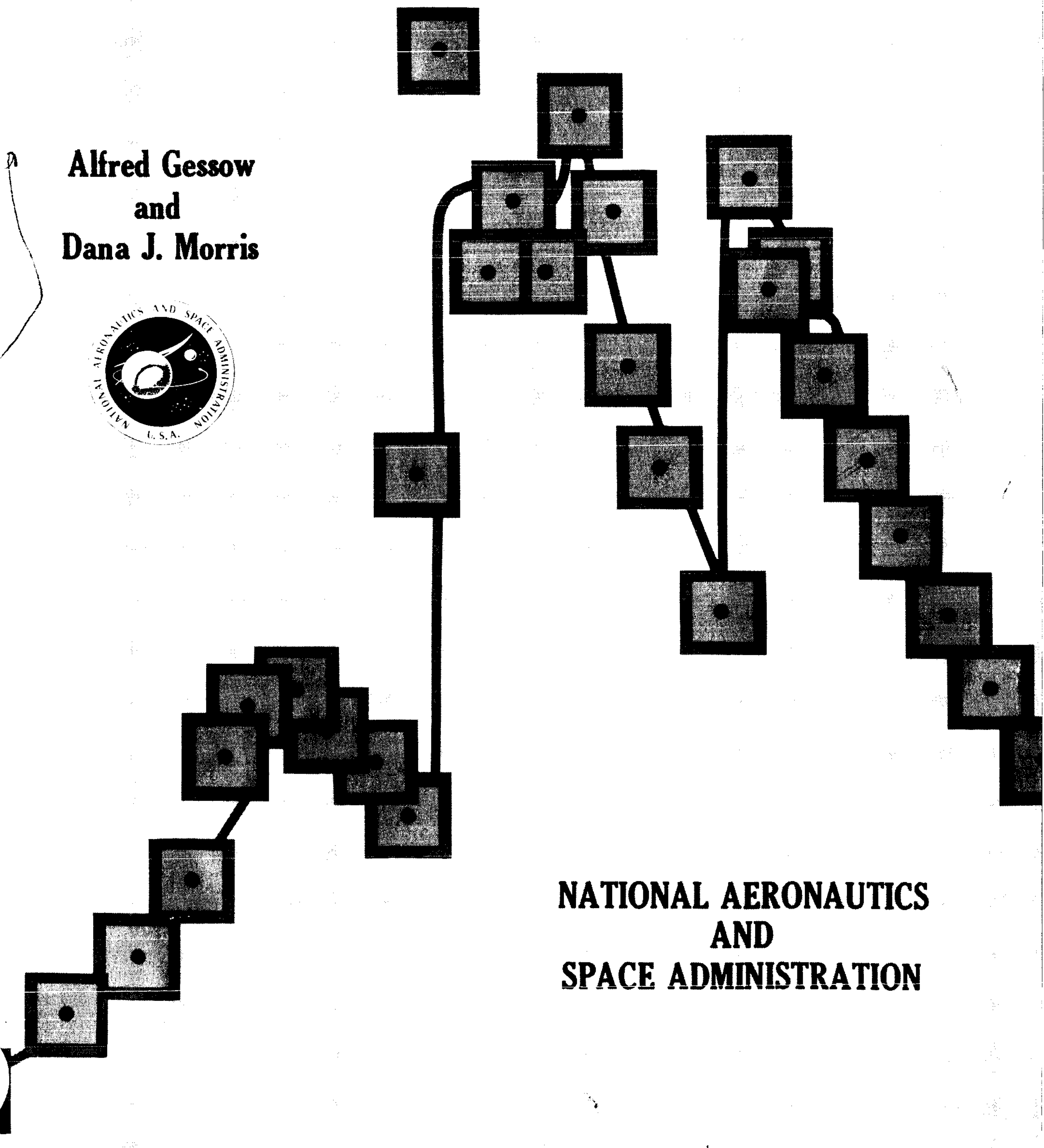
NASA SP-394

A Survey of Computational Aerodynamics in The United States

Alfred Gessow
and
Dana J. Morris



NATIONAL AERONAUTICS
AND
SPACE ADMINISTRATION

NASA SP-394

A Survey of Computational Aerodynamics in the United States

Alfred Gessow
and
Dana J. Morris

Scientific and Technical Information Office 1977
NATIONAL AERONAUTICS AND SPACE ADMINISTRATION
Washington, D.C.

PREFACE

This survey paper was presented to the AGARD Fluid Dynamics Panel in October 1975 to aid the panel in formulating plans for a future conference on theoretical and computational aerodynamics. It describes in part programs in theoretical and computational aerodynamics in the United States. It details only those aspects of those programs that relate to aeronautics and not the related development effort involving application of aerodynamic theory to such problems as planetary entry and atmospheric and oceanographic modeling. The role of analysis at various levels of sophistication is discussed as well as the inverse solution techniques that are of primary importance in design methodology. The research discussed is divided into the broad categories of application for boundary layer flow, Navier-Stokes turbulence modeling, internal flows, two-dimensional configurations, subsonic and supersonic aircraft, transonic aircraft, and the Space Shuttle. A survey of representative work in each area is presented, with the exception of turbulence modeling techniques, which were covered in a recent AGARD presentation.

CONTENTS

INTRODUCTION

The role of computational aerodynamics is to aid in advancing technology of long-range aircraft systems, helicopters and rotorcraft, general aviation aircraft, and maneuverable fighter aircraft and missiles with the generation of design and analysis methods that provide improved definition and lower costs in developing such vehicles. The development of new aerodynamic concepts evolves from a clear understanding of the physics of flow fields. To obtain this understanding with the present state of the art, an appropriate balance between theory and experiment must be maintained at all research levels. Although new design and analysis methodology may take the form of computational techniques, its development also rests squarely on the dual foundations of theory and experiment.

Fluid flow research has generally emphasized the complementary roles of theory and experiment. For a number of years, the experimentalists directed most of their effort toward determining aerodynamic characteristics for point design and obtaining systematic data for configurations and flow phenomena not yet amenable to analysis. Experimental programs for validating a theoretical method or for providing data for the generation of a theoretical model were few. Theoretical methods were based primarily on linear, incompressible, inviscid flow equations or on the Prandtl boundary layer equations and were applicable to only the simplest configurations. With the formulation in the late 1940s and 1950s of a number of methods for solving approximate linear and exact forms of the governing equations for the flows over practical geometries, an increased emphasis on experimental research for theory validation took place. This trend has accelerated in recent years, stimulated by a rapidly improving analysis capability, which may be attributed to growth in the speed and storage capacity of electronic computers as well as the development of improved numerical solution techniques.

In 1946 John von Neumann described wind tunnel testing as follows (ref. 1):

> Indeed, to a great extent, experimentation in fluid dynamics is carried out under conditions where the underlying physical principles are not in doubt, where the quantities to be observed are completely determined by known equations. The purpose of experiment is not to verify a theory but to replace a computation from an unquestioned theory by direct measurements. Thus wind tunnels, for example, are used at present, at least in part, as computing devices to integrate the partial differential equations of fluid dynamics.

As computer simulations become more feasible because of rapidly improving computer capability, the need for as much wind tunnel testing will decrease. The dual partnership of theory and experiment continues; however, their roles in relation to aeronautics programs are beginning to change with the expanding role of the computer that has occurred because of its increasingly accurate simulations, as indicated in figure 1. The requirement for point design experimental data has steadily diminished as numerical methods have improved. Limitations imposed by most experimental programs, where only a few configurations can be tested over a limited range of conditions, are being circumvented to an increasing extent by emphasizing tests specifically designed to check theory at a few critical points or to generate and improve theoretical flow models. Once an analysis is proven adequate, it provides an interpolative and extrapolative capability for handling other geometries or flow conditions for which test data do not exist. As the cost of wind tunnel testing increases, computer simulations are expected to reduce the time and cost of

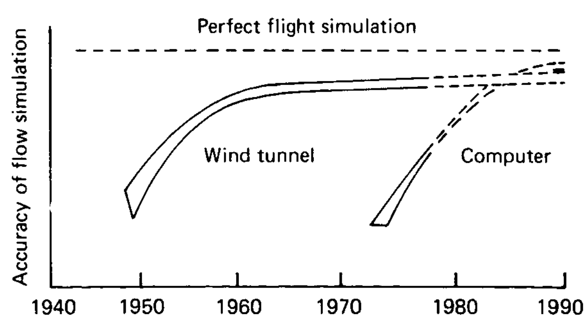


Figure 1.–Comparison of accuracy of wind tunnel testing and computer simulations.

developing new prototype vehicles. In summary, then, computational fluid dynamics has the following advantages over wind tunnel testing:

(1) It provides the capability of simulating flow impossible or impractical to simulate in a ground-based experimental facility.

(2) It provides more accurate simulations of flight aerodynamics than wind tunnels can provide.

(3) It reduces the time and cost of obtaining flow simulations necessary in the design and analysis of new prototype aerospace vehicles.

In the 1980s computer methods and wind tunnel testing will perform complementary functions.

New numerical techniques and large computers have also allowed solutions of problems that, until a few years ago, could not have been attempted even with the most gross assumptions. Solutions of the Navier-Stokes equations, or slightly simplified versions, for viscous flow over multielement airfoils, shock/boundary-layer interactions, various jet-exhaust and free shear flows, and for a variety of two-dimensional geometries where flow separation is an important feature, indicate a new level of technology that will mature swiftly with the development and use of computer systems such as the CDC STAR or the ILLIAC IV. With the use of such or even more advanced computers, a similar trend may take place in the application of numerical simulation to flows over complex aerospace configurations.

METHODS AND STATUS

The basic process used in theoretical and computational aerodynamics is the solution of a set of governing partial differential equations, which are known, under constraints imposed by the initial and boundary conditions specified as appropriate to the problem at hand. The representations (mathematical model) of these equations and boundary conditions of the physical problem are determined by the choice of numerical method, computer language, and by the machine itself. A computer code is then developed to provide a solution to the modeled problem. Results of calculations from the code are compared with experimental data to determine the validity of the representation. Experimental data are also needed to provide empirical inputs for the computer code, and, as computations are becoming available for more sophisticated flow fields, researchers are able to determine from computed solutions areas in which experiments will be particularly helpful in understanding aerodynamic phenomena. Thus the cycle of experiment and analysis illustrated in figure 2 continually advances both efforts.

The time-dependent Navier-Stokes and energy equations are generally accepted as a valid set of governing equations for aerodynamic flows without chemical reactions. In 1972 Bradshaw stated (ref. 2),

> In turbulence studies we are fortunate in having a complete set of equations, the Navier-Stokes equations, whose ability to describe the motion of air at temperatures and pressures near atmospheric is not seriously in doubt. (It is easy to show that the smallest significant eddies are many times larger than a molecular mean free path.) We are unfortunate because numerical solution of the full time-dependent equations for turbulent flow is not practicable with present computers.

Because the Navier-Stokes equations are highly nonlinear, their exact solution is virtually impossible. Useful solutions in aerodynamic theory have been made possible by the application of simplifying approximations of one form or another to the equations and their boundary conditions. Some standard

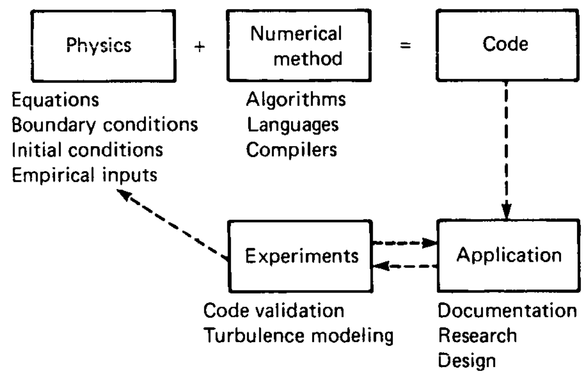


Figure 2.—Elements of a computational fluid dynamics program.

approximations, using the x momentum equation alone for illustration, follow, in increasing order of approximation. The x momentum equation is

$$\rho\left(\frac{\partial u}{\partial t} + u_j \frac{\partial u}{\partial x_j}\right) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial u}{\partial x_j}\right) - \frac{2}{3}\frac{\partial}{\partial x}\left(\mu \frac{\partial u_j}{\partial x_j}\right) + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial u_j}{\partial x}\right)$$

In the Reynolds model, velocity is assumed to be the sum of an average and a fluctuating quantity; i.e.,

$$u_j = \bar{u}_j + u_j'$$

therefore,

$$\bar{\rho}\left(\frac{\partial \bar{u}}{\partial t} + \bar{u}_j \frac{\partial \bar{u}}{\partial x_j}\right) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x_j}\left(\bar{\mu} \frac{\partial \bar{u}}{\partial x_j}\right) - \frac{2}{3}\frac{\partial}{\partial x}\left(\bar{\mu} \frac{\partial \bar{u}_j}{\partial x_j}\right) + \frac{\partial}{\partial x_j}\left(\bar{\mu} \frac{\partial \bar{u}_j}{\partial x}\right) - \frac{\partial}{\partial x_j}\overline{[(\rho u_j)'u']}$$

In boundary layer modeling, it is assumed, in addition to the Reynolds assumptions, that

$$\bar{u} >> \bar{v}$$

and

$$\frac{\partial \bar{u}}{\partial x} >> \frac{\partial \bar{v}}{\partial x}$$

therefore,

$$\overline{\rho u}\frac{\partial \bar{u}}{\partial x} + \overline{\rho v}\frac{\partial \bar{u}}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial y}\left(\bar{\mu}\frac{\partial \bar{u}}{\partial y} - \overline{\rho v'u'}\right)$$

The inviscid nonlinear model assumes $\mu = 0$ in the Navier-Stokes equations, thus

$$\rho\frac{\partial u}{\partial t} + \rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x}$$

After definition of a perturbation velocity and some algebraic manipulation, this equation becomes

$$(1 - M_\infty^2)\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = M_\infty^2(\gamma - 1)\frac{u}{U}\frac{\partial u}{\partial x}$$

In the inviscid linearized model, the right-hand side of the preceding equation is neglected; therefore,

$$(1 - M_\infty^2)\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

Each form of approximation or subdivision thereof leads to its own solution techniques with comparative advantages and disadvantages with respect to generality of application, accuracy of results, and cost of solution. Early computers were capable of computations simulating only the simplest flows. As computer storage and speed have increased, the level of approximation that could be simulated has increased. Table 1 lists the principal limitations in application, the status of available computer codes, and the pacing items in code development for various

TABLE 1.–*Stages of Approximation to Governing Equations of Fluid Dynamics*

Characteristics	Approximations	Principal limitations	Status	Pacing item
Inviscid linearized	Viscous and nonlinear inviscid terms neglected	Slender configurations; small angle of attack; perfect gas; no transonic flow; no hypersonic flow; no flow separation	2-dimensional flows in 1930s; 3-dimensional flows in 1960s; used in current aircraft design	–
Inviscid nonlinear	Viscous terms neglected	No flow separation	2-dimensional flows in 1971; 3-dimensional flows in 1975; early stages of application to aircraft design	Code development
Viscous boundary layer	Streamwise velocity gradients neglected; turbulence terms modeled by eddy viscosity	No flow separation; no large crossflow; no large pressure gradients	2-dimensional flows in 1971; 3-dimensional flows in 1974; early stages of application to aircraft design	Code development
Viscous time-averaged Navier-Stokes	No terms neglected; turbulent momentum, energy, and heat transport terms modeled	Accuracy of turbulence model	2-dimensional flows in 1975; 3-dimensional flows possibly by 1978; early stages of development with simple turbulence models for 2-dimensional flows	Development of improved turbulence models
Complete viscous time-dependent Navier-Stokes	Subgrid scale motion modeled	Accuracy of Navier-Stokes equations	Mid-1980s	Development of advanced computer

TABLE 2.–*Methods of Solution*

Method	Advantages	Disadvantages	Comments
Explicit finite differences (for example, MacCormack, hopscotch, and DuFort-Frankel)	Easy to code; incorporation of turbulence modeling is straightforward	Large number of nodes and long computation times	Most frequently used in the past because of simplicity
Implicit finite difference (for example, alternating direction implicit (ADI))	Potential for shorter computation time	Difficult to code; requires matrix reduction; requires excessive storage	Advent of large computers has made this feasible
Finite element	Same degree of accuracy requires fewer nodes; easily adapted to irregular geometry; derivative boundary conditions incorporated directly	Data handling complex and requires excessive time and storage	Widely used in structures; now being investigated for fluid problems
Cubic spline integration	Same advantages as finite element; handles shock waves easily	Requires tridiagonal matrix reduction	Recently developed technique that still has to prove itself in application

stages of approximation to the Navier-Stokes equations.

As noted previously, the equations represent the physical problem. The numerical procedure chosen to model the equations provides the mathematical representation of this problem in the computer. Listed in table 2 are some of the procedures that are currently used, and the principal advantages and disadvantages of each method. Because the computation cost and storage requirement are a direct function of the complexity of the equation simulated, the simplest approximation of the governing equations that is applicable to the physical problem should be used. Once the chosen equations are cast in a finite difference form by using a numerical technique, the solution can proceed either by a time marching or a relaxation method. At present, there is no one optimal numerical technique. The choice of the numerical method must be based on the flow characteristics of a particular application and the operating characteristics of the available computational facility. In the case of numerical simulation of aircraft configurations–the shuttle configuration shown in figure 3, for example–highly integrated and specialized applications of several solution techniques are required depending on the level of detail necessary and the location on the vehicle where the flow is of interest.

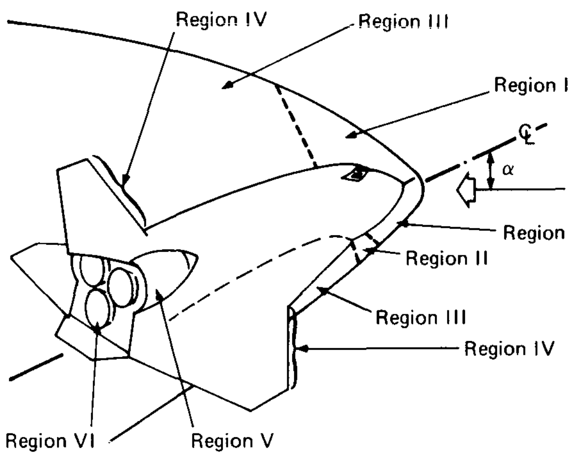


Figure 3.–The combination of numerical procedures used in the analysis of a single configuration. Region I: subsonic flow; must march in time. Region II: supersonic flow; no plane normal to body axis with all supersonic flow; use space marching mapped coordinates. Region III: supersonic flow; planes normal to body axis with all supersonic flow do exist; use parabolic Navier-Stokes equations with space marching body coordinates. Region IV: subsonic flow; embedded regions of subsonic flow around wing and tail leading edges; use full Navier-Stokes time marching or couple potential equations with boundary layer equations. Region V: internal flow; reacting chemistry must be included. Region VI: nozzle exhaust; three-dimensional mixing; shock interacting.

SURVEY

A survey of representative U.S. work in computational aerodynamics is presented in tables 3 to 8 in terms of the categories of boundary layer flow, internal flows, two-dimensional configurations, subsonic and supersonic aircraft, transonic aircraft, and space shuttle. Highlights in each category are discussed and illustrated with specific examples in the following sections. The large amount of effort directed toward Navier-Stokes turbulence modeling is not included in tables in this document because it is the subject of a separate AGARD review (ref. 3). Some discussion of this effort is presented, however, because of its importance to the future of the entire computational aerodynamics effort.

VISCOUS FLOWS

Viscous flow aerodynamics is discussed prior to and apart from the internal flow and configuration categories because of its importance and across-the-board applicability. Work in this category divides itself naturally into the areas of boundary layer flows and Navier-Stokes solutions.

Boundary Layers

Numerical solutions for two-dimensional and three-dimensional laminar boundary layers over simple shapes without large adverse pressure gradients can presently be achieved to any desired accuracy. With existing models for the Reynolds stresses, turbulent boundary layer flows can be calculated with reasonable accuracy. There has also been some success in computing 2-dimensional flow through small embedded separation regions using the boundary layer equations; however, 3-dimensional flow near separation is not fully understood. Although prediction methods for laminar and turbulent flow are accurate, little is known about the prediction of transition and relaminarization. Current research is concentrating in these areas and in developing more efficient procedures. A sampling of this research is given in table 3. No distinction is made in the table between codes that are for research only and those that are or will be available as production codes.

There have been several survey papers on solution procedures for the boundary layer equations. Also, books are available describing some of the methods in detail. Second-order accurate, implicit and explicit finite difference schemes exist for solving steady two-dimensional and axisymmetric perfect gas flow. The method of weighted residuals, the matrix integral method, and higher order finite difference methods have also been applied with success. Optimal spacing of computational nodes normal to the wall boundary plays a vital part in retaining the desired accuracy while minimizing computer time for turbulent flows. As more efficient techniques are developed, they can be used in unsteady, three-dimensional, and real-gas flows. Some two-dimensional techniques have been extended to the unsteady flow case. Additional work on the appropriate transformation to use with the unsteady boundary layer equations is needed.

For steady three-dimensional flows, difference schemes have been developed and problems have been solved for various initial and boundary conditions. One problem appears to be the development of a general code for solving a variety of three-dimensional flows. Such a development requires further evaluation of coordinate systems, transformations, and more accurate and better ways of handling the required inviscid flow data. Flexibility needs to be added into the codes so that the various difference schemes can be automatically used as required to satisfy zones of dependence. There appear to be no new problems in extending three-dimensional numerical techniques to unsteady flows, other than computer storage requirements, unless reverse flow is encountered. Some three-dimensional solution procedures have been extended to turbulent flows with no more difficulty than encountered for two-dimensional extension to turbulent flows. One implicit code used by a number of investigators employs a second-order differencing scheme that is conditionally stable for reverse crossflow. It has an option for either two-layer eddy vis-

TABLE 3.–*Boundary Layer*

Investigator	Method	Application	Code status
J. Adams,	Implicit finite difference	Yawed airfoil	Operational for 3 dimensions
H. Dwyer	Finite difference, hybrid scheme	Spinning cone	Operational for 3 dimensions
J. Nash	Differential method	Airfoil sections	Operational for 3 dimensions
A. J. Baker	Finite element	Combustors; duct flows	Operational for 3 dimensions
K. C. Wang	Mixed one- and two-step finite difference	Blunt body	Operational for 3 dimensions
A. Wortman	Differential operator technique	Blunt body; swept and tapered wings	Operational for 3 dimensions
R. T. Davis	Implicit finite difference	Sharp and blunted cones	Operational for 3 dimensions
F. Blottner	Finite difference	Analytic bodies	Operational; plans include incorporation of flexible integration direction
R. M. Kendall and M. Abbett	Implicit solution: finite difference and splines	Windward surface with real gas effects	Operational; plans include coupling with 3-dimensional inviscid code
T. Cebeci	Keller "box" method	Perfect gas analysis for wings of arbitrary shape	Operational with eddy viscosity model; plans include coupling with 3-dimensional inviscid code
J. Harris	Implicit finite difference	Analytic bodies; option for viscosity model	Analysis operational for 3 dimensions; plans include extension to general geometry
A. J. Baker	Finite element	Reentry vehicle in real gas	Operational for 2-dimensional supersonic flow; plans include extension to subsonic flow
M. Frieders and C. Lewis	Implicit finite difference	Sharp and blunt cones with real gas effects	Operational for 3 dimensions

cosity or mixing length turbulence models. Investigations have been made concerning reduction in nodal spacing to optimize computer time and minimize storage. Some typical results presenting calculations using the two different turbulence models and two nodal spacings are shown in figure 4 (from ref. 4). A typical case of 101 points in the normal direction requires 300 s and 70 000 octal storage locations on a CDC 6600 machine. An example of the capability of the current analysis at hypersonic speeds to handle real-gas effects on a blunted cone at angle of attack is shown in figure 5 (from ref. 5). The good agreement on the leeward side is not surprising because of the relative small angle of attack of the surface.

Navier-Stokes and Turbulence Modeling

For viscous flows where the boundary layer assumptions are invalid, the full time-dependent Navier-Stokes equations must be solved. Such solutions are presently obtainable for only a restricted class of these flows because of limitations in the finite difference computations. Turbulent flow is particularly difficult to calculate. At a point in space, turbulent flow is irregular and random. It is a three-dimensional phenomenon composed of different structures having a wide range of time and length scales. To compute these irregularities, accurate time-dependent computational techniques must be used. Numerical stability and accuracy restraints on the allowable time step are imposed by the small-scale dimensions required to prevent errors in the instantaneous flow field from growing. These requirements for computer speed and storage capacity are beyond available and projected computer capabilities; thus it is necessary to resort to turbulence modeling.

The first step in turbulence modeling is the elimination of the small-scale structure of the dependent variable by an averaging process (ensemble, space, or

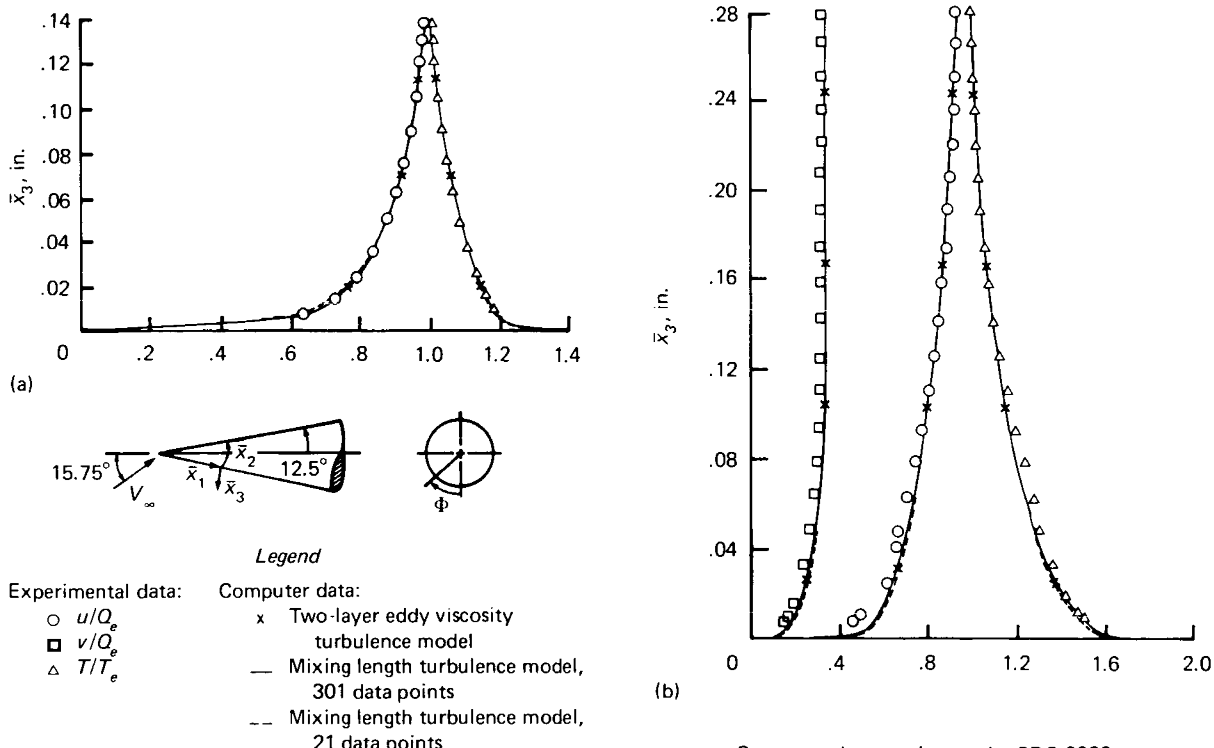


Figure 4.—Experimental data and computer calculations for two different turbulence models at two nodal spacings (from Harris and Morris, ref. 4). $M_\infty = 1.8$; $\mathrm{Re}_{\infty,L} = 25 \times 10^6$; $\gamma = 1.4$; $Q_e = (u_e^2 + v_e^2)^{1/2}$. (a) $\Phi = 0°$. (b) $\Phi = 135°$.

Storage and processing on the CDC 6600

Number of data points	Central processing unit mesh point, s	Storage	Test case time, s
301	0.0123	170 000	900
21	.0123	60 000	60

time). The new averaged dependent variables can be resolved with finite difference methods using mesh spacing and time intervals compatible with present computer capability. The nonlinear terms in the basic equations introduce averages of products of the small-scale portions of the dependent variables that must be related to the averaged values (the "closure" problem). This "closure" is the second step in the modeling process. Here much emphasis is placed on comparisons with experimental data because the modeling is empirical.

One method under investigation, suitable for steady-state mean flow fields, is called statistical theory of inhomogeneous turbulence, where averaging is performed at a point in space over a long period of time compared with the time scales of the largest turbulent eddies. The closure is achieved by eddy viscosity modeling. An example of the effectiveness of the eddy viscosity turbulence model in transonic flow as applied to a circular arc airfoil is given in figure 6 (from ref. 7). The model is shown to be quite effective when compared with the inviscid calculation in the region of attached flow or when the separation is modest as occurs in trailing edge separation. With shock-induced separation it is clear that the model used is inadequate although the major features of the flow are qualitatively predicted. Figure 7 (from ref. 8) compares the experimental results of a well-documented experiment in which a shock wave was used to separate a turbulent boundary layer at mach 7.2. The predicted quantities were calculated with a statistical turbulence model of the Navier-Stokes equations. Although the general features of the data are computed, there are significant differences that require improvement. The dashed lines represent an improved model in which the data were analyzed to change the principal modeling constants. Some improvements occur, especially with regard to the upstream influence, but the algebraic models are most limited and much needs to be done to improve the

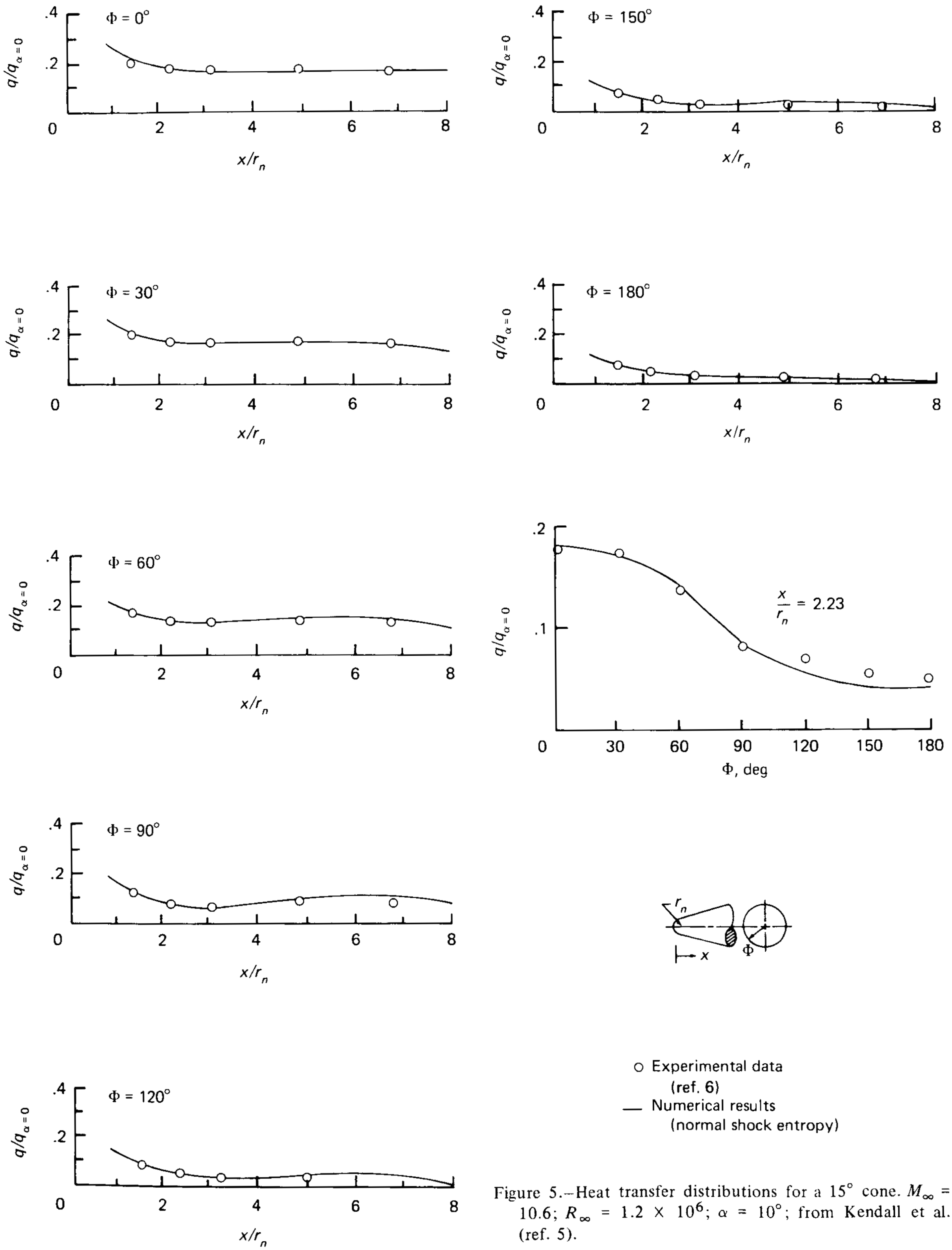


Figure 5.—Heat transfer distributions for a 15° cone. M_∞ = 10.6; R_∞ = 1.2 × 10^6; α = 10°; from Kendall et al. (ref. 5).

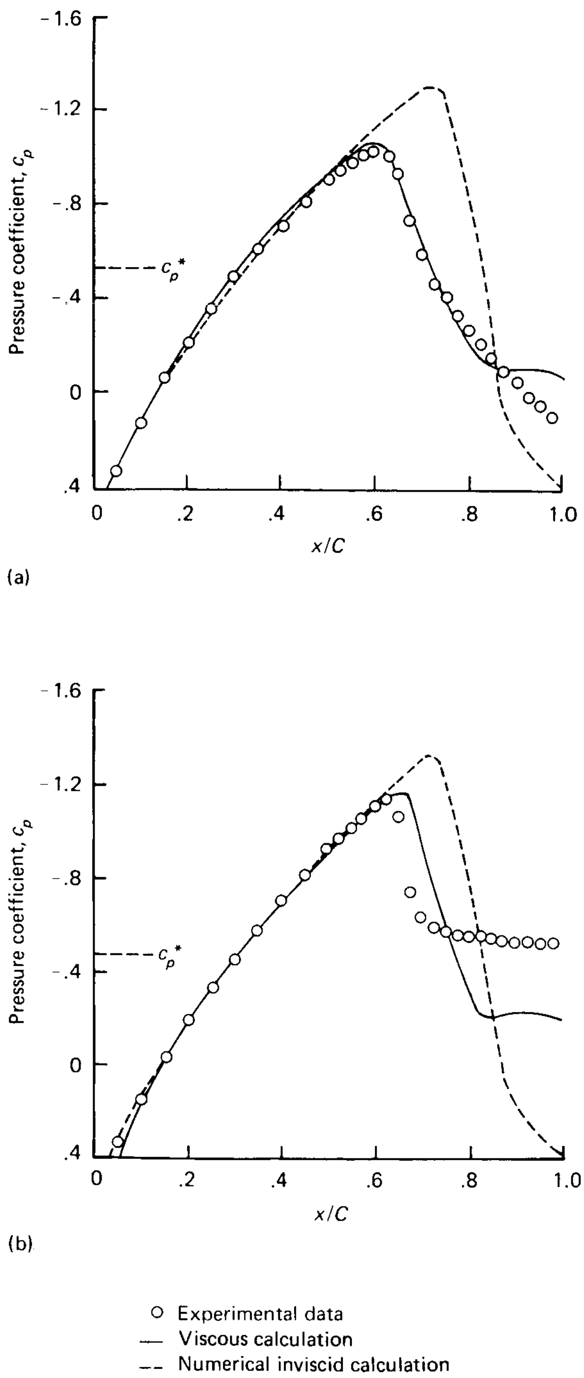


Figure 6.—Numerical solution of circular arc airfoil using Navier-Stokes equations with eddy viscosity turbulence model (from Deiwert, ref. 7). (c_p* = critical pressure coefficient.) (*a*) Trailing edge separation. M_∞ = 0.775; $\mathrm{Re}_{c,\infty}$ = 2 × 10^6. (*b*) Shock-induced separation. M_∞ = 0.786; $\mathrm{Re}_{c,\infty}$ = 10^7.

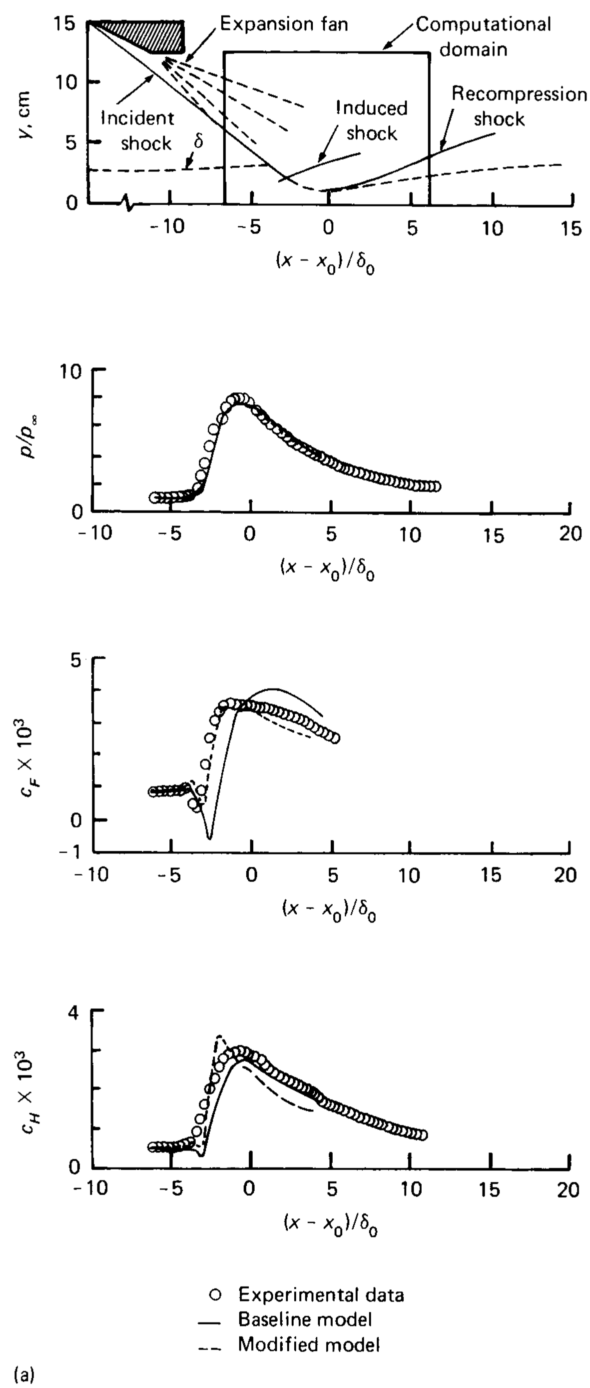
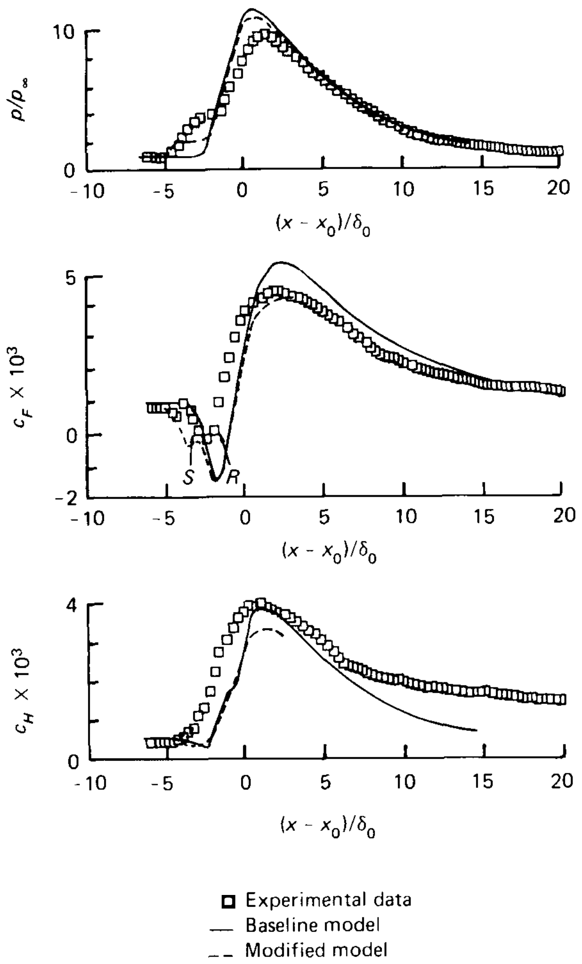


Figure 7.—Pressure, skin friction, and heating calculations for oblique shock wave/turbulent boundary layer interaction at mach 7.2 (from Marvin et al., ref. 8). (c_F = skin friction coefficient; c_H = heat transfer coefficient.) (*a*) α = 7.5°.



(b)

Figure 7 (concluded).—(*b*) α = 15°; *S* = separation; *R* = attachment.

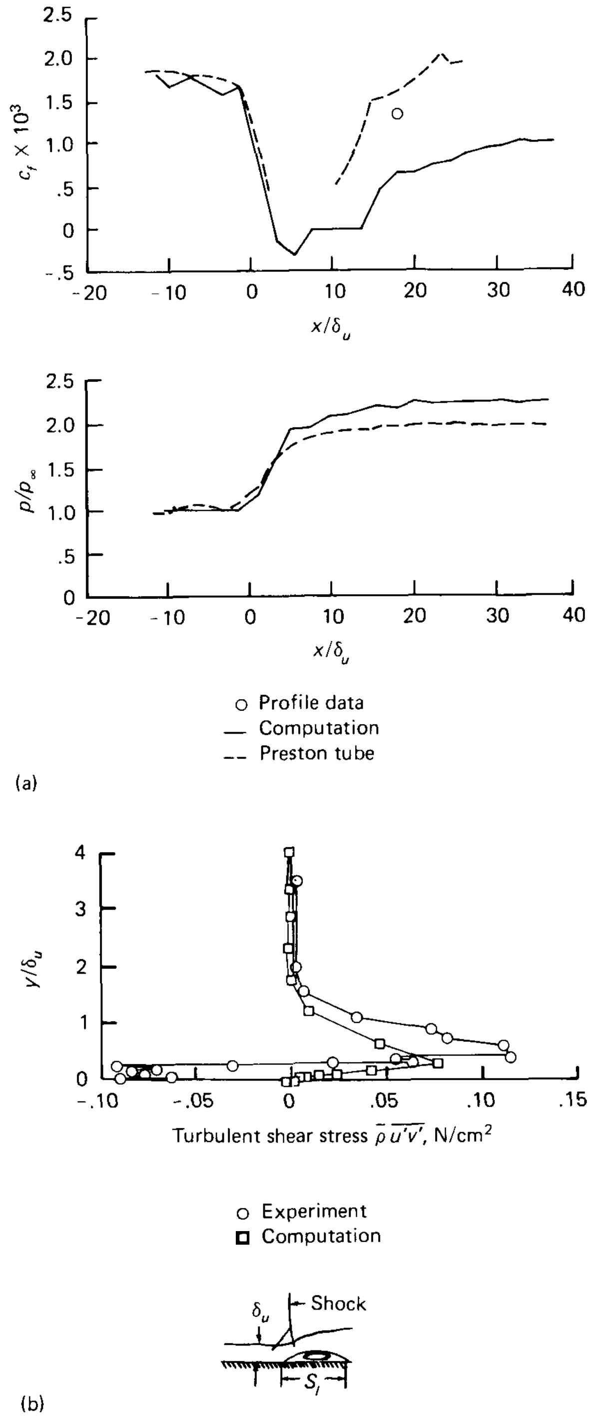


Figure 8.—Normal shock wave/turbulent boundary layer interaction in a circular channel at mach 1.5 (from Mateer, Brosh, and Viegas, ref. 9). (*a*) Comparison of wall pressure and skin friction. (*b*) Normal shear stress distribution; x/δ_u = 4.

modeling for this type of a complex flow field. Another example of the effectiveness of the eddy viscosity model is shown in figure 8 (from ref. 9) for the case of a normal shock wave turbulent boundary layer interaction at mach 1.5. The numerics adequately predict the skin friction up to separation but underestimate skin friction downstream of reattachment. However, the length of the separated flow region is fairly closely predicted. Also, although the simulation considerably underpredicts the height of the reverse flow region, it does provide a good qualitative description of the flow by identifying the extent of the shear layer and the location of maximum shear. Ultimately, a prediction method may have to drop the eddy viscosity concept and calculate the Reynolds stresses directly.

An alternative method, sometimes called "turbulent simulation," is based on averaging over space volumes that are smaller than the largest turbulent eddies but much larger than the smallest eddies. The volume averages retain their time dependence but possess scales resolvable by current computer techniques. Turbulent simulation is expected to become the basis for practical turbulence computation because inherent in its formulation is its direct evaluation of the large eddies that are characteristic of a particular flow and its modeling of the small "subgrid" scales that are thought to be universal in character (fig. 9).

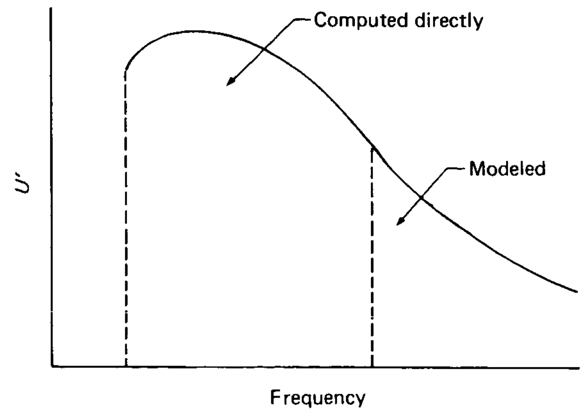


Figure 9.–Goal of turbulence modeling with projected computer capability. Subgrid-scale turbulence modeling is used to close the system of equations.

INTERNAL FLOWS

Aircraft with airbreathing propulsion require a high degree of engine/airframe integration to achieve optimized performance. Although the complex flow interactions resulting from integrated aircraft configurations present a formidable challenge to analysis, the effects of these interactions must be determined to produce an effective aircraft design. Representative research in this area is listed in table 4.

Inlets

Inlet flows represent one of the more difficult challenges in the development of computational techniques. For the most part, inlets have been developed through a combination of empiricism and many hours of wind tunnel testing. Analytical techniques are needed for the detailed design of internal contours, mass removal systems, and for future aircraft to blend the propulsion system into the

TABLE 4.–*Internal Flows*

Investigator	Method	Application	Code status
D. A. Oliver and P. Sparis	Finite difference, inviscid	Turbomachinery	Operational
J. Erdos and E. Alzner	Finite difference, inviscid	Turbomachinery	Operational
L. L. Presley	Finite difference, shock-capturing, inviscid	Supersonic inlet	Operational
P. Kutler and V. Shankar	Finite difference, inviscid	Integrated ram/scram jet nacelles	Operational
S. M. Dash and P. D. Del Guidice	Finite difference	3-dimensional nozzle exhaust flow hydrogen-air chemistry	Operational
C. du P. Donaldson	Finite difference, second-order closure	Combustion in scram jets at low mach numbers	Operational
R. S. Hirsh	Finite difference (ADI), x-marching, viscous	3-dimensional nozzle exhaust	Operational for supersonic flow; plans include extension to subsonic
H. McDonald and W. R. Briley	Finite difference (ADI), x-marching, viscous	Internal flows	Operational
W. S. Llewellen and R. D. Sullivan	Finite difference, x-marching, viscous	Subsonic wake flow now being applied to vortex mixing	Operational
M. D. Salas	Finite difference (MacCormack) and floating shock fitting, inviscid	Supersonic inlet	Operational

overall configuration with minimum penalty to, or perhaps enhancement of, the basic aircraft performance.

Most of the analytical work in the past has been applied to the development of computer codes for planar and axisymmetric supersonic flows using the method of characteristics. Pressure distributions from these codes are used as input to viscous codes incorporating engineering models of shock-wave boundary-layer interactions to predict the extent and magnitude of boundary layer removal.

Solution of subsonic and transonic internal flows is not as well developed as supersonic flows. Approaches to subsonic flows have usually resulted from incompressible potential flow solutions, with compressibility corrections being applied. Transonic inlet flows have been approached using small disturbance theory as well as time-dependent solutions of the equations of motion. Significant progress has been made in analyzing subsonic duct flows in which viscous effects and arbitrary cross section are included in the analysis. Recent developments have been along several lines:

(1) Developing techniques for full three-dimensional supersonic flows, including corners

(2) Developing three-dimensional techniques for transonic flows

(3) Developing improved viscous techniques for analyzing flows with large adverse pressure gradients and mass removal

(4) Developing analytical techniques that simultaneously analyze the inviscid and viscous portions of the flow and include coupling effects

One method is illustrated in figure 10 (from ref. 10) where calculations of pressure ratio in the inlet are compared with a solution by the method of characteristics.

It should be emphasized that the primary emphasis of future efforts should be in developing analytical techniques that can be applied to general three-dimensional configurations and that viscous effects should be included. However, throughout the development of new computational techniques, continual comparison with experimental standards must be made to verify the accuracy and physical modeling of the new techniques.

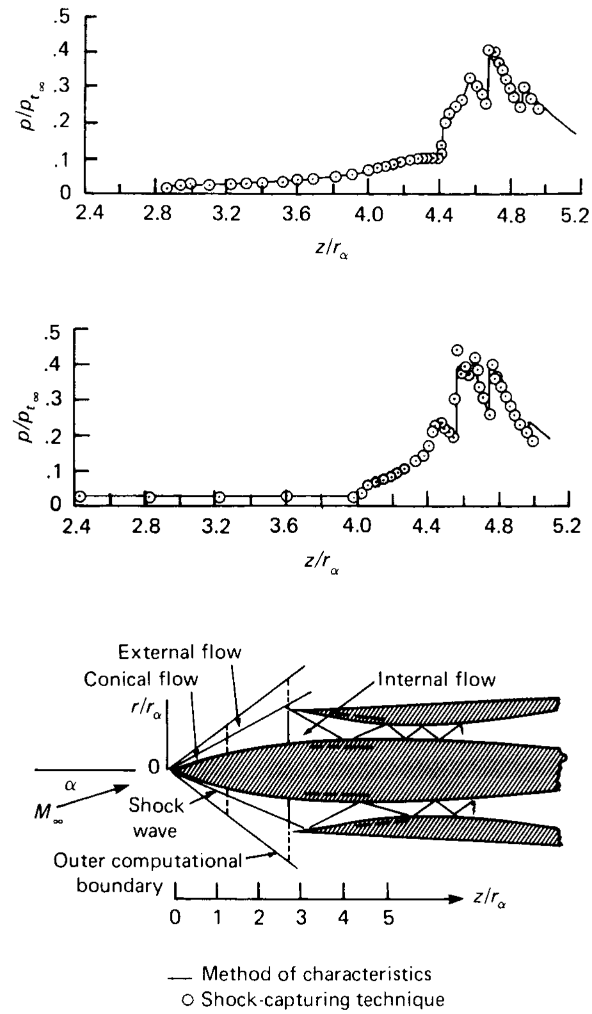


Figure 10.–Comparison of numerical techniques (from Presley, ref. 10). M_∞ = 3.5; α = 0°. Dashed lines indicate region of boundary layer removal.

Turbomachinery

The flow field within advanced axial turbomachinery, e.g., high bypass ratio turbofan engines, is characterized as highly three-dimensional and unsteady, with transonic to supersonic mach numbers and multiple shocks. Conventional analytical methods place reliance upon two-dimensional potential flow and linearized three-dimensional flow models, and, therefore, constrain to some extent the design and analysis of new systems. An example of the development of more advanced methods of aerodynamic analysis based on numerical solution of the complete equations of motion for an unsteady inviscid compressible flow is the joint effort undertaken by NASA Lewis Research Center and by Advanced Technology Laboratories, Inc. The programs employ a shock-capturing finite-difference method at the interior grid points and a reference plane method-of-characteristics procedure at the boundary points. Subsonic axial velocity is assumed at both the inlet and discharge stations, and either choked or unchoked operation can be considered. One program pertains to analysis of meridional hub-to-shroud stream surfaces and another to axisymmetric blade-to-blade stream surfaces (fig. 11, from ref. 11). Both programs can analyze either one or two blade rows (i.e., rotor and/or stator), and the meridional program can consider multiple stages. Finite blade thickness, camber, and lean angles are included. In the meridional program, a steady, axisymmetric solution with crossflow (i.e., swirl) is obtained. In the blade-to-blade program, a steady two-dimensional solution is ob-

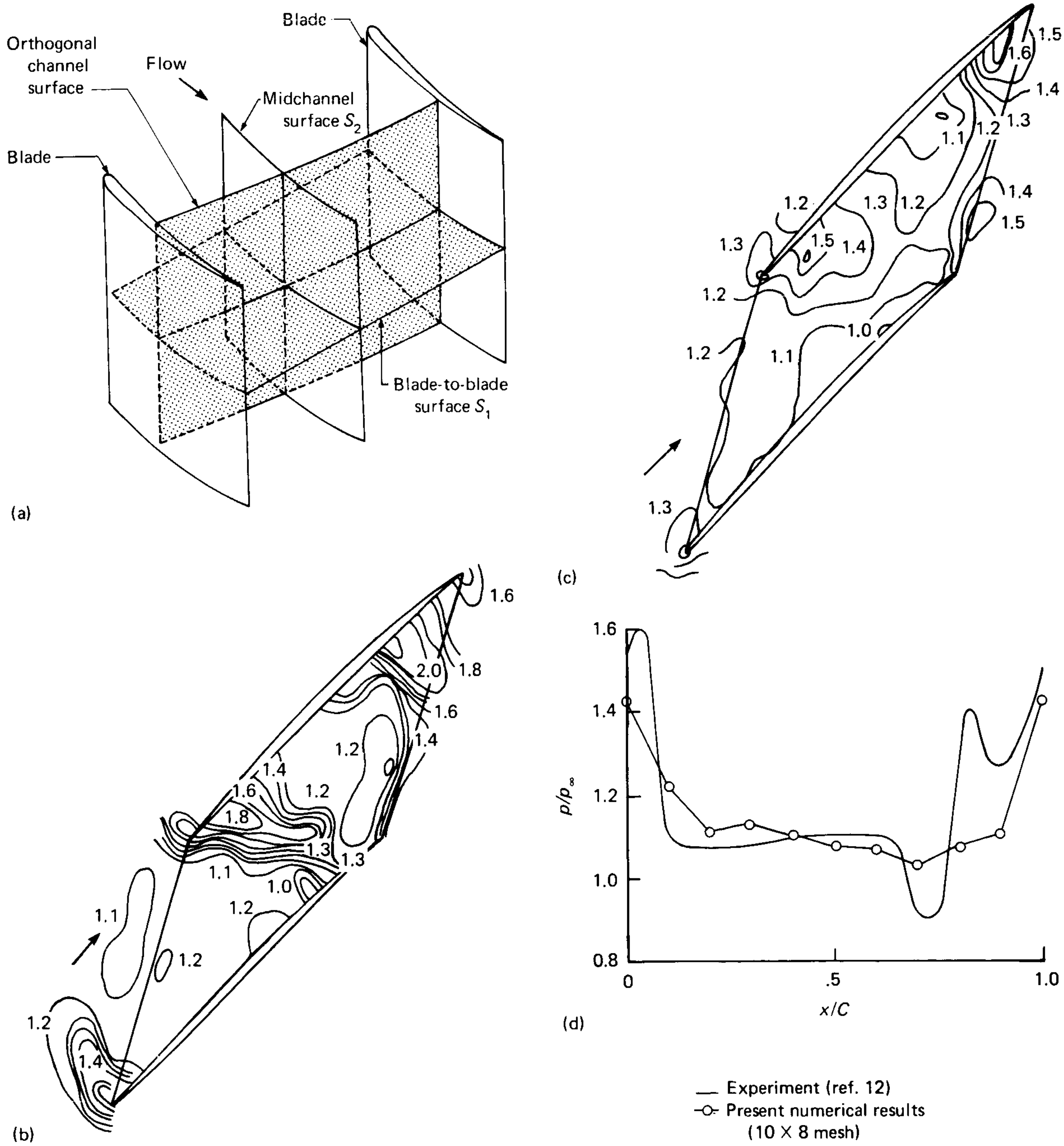


Figure 11.–Rotor pressure distributions (from Erdos et al., ref. 11). (*a*) Diagram. (*b*) Experimental isobars (ref. 12). (*c*) Numerical isobars. (*d*) Suction surface.

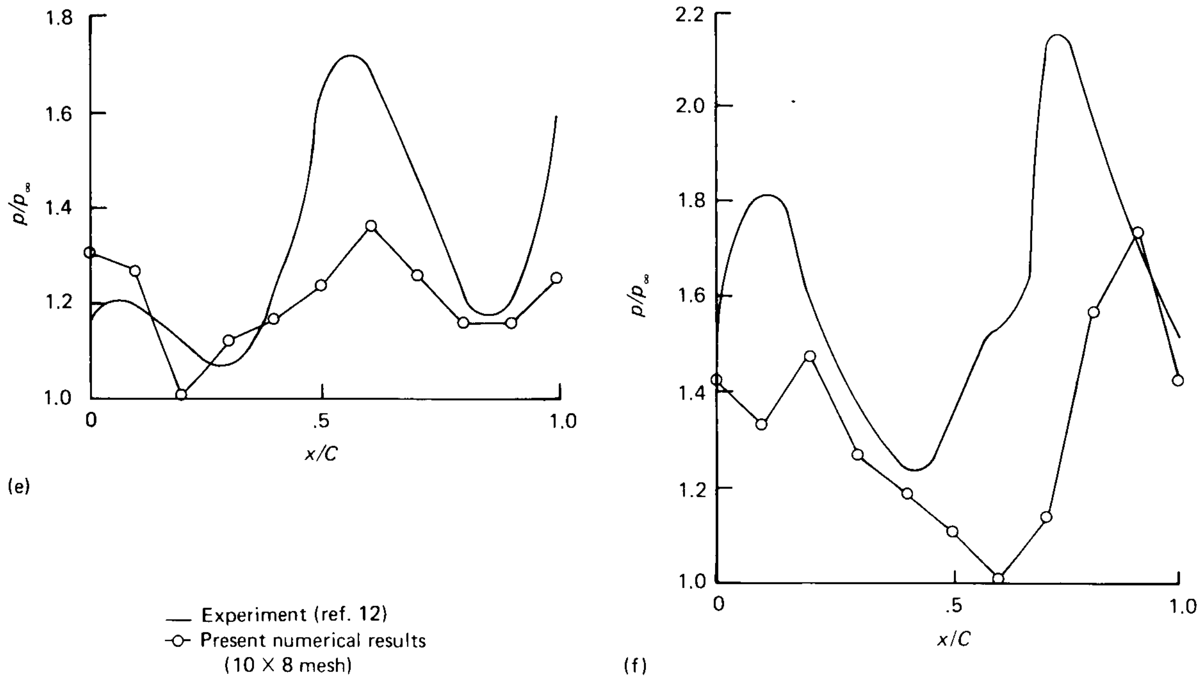


Figure 11 (concluded).–(*e*) Midchannel line. (*f*) Compression surface.

tained for a single blade row, and a periodic unsteady flow is obtained with a rotor and stator. Numerical results to date consist of certain test cases for a single transonic rotor blade row (for which experimental data are available) and a hypothetical rotor/stator combination. The computed results agree favorably with experimental data, as shown in figure 11, in which the free stream mach number relative to the rotor is 1.5. The present computer program produces a reasonably good rendition of the experimental data after 200 time steps, which required only 140 s of CDC 6600 computer time.

Combustor

The complex flow field resulting when two or more ducts merge, as, for example, the flow produced by fuel injectors in a nozzle or the merging flow of a series of scramjet nodules, produces a number of interacting shocks. A finite difference computer code has been developed to analyze the flow of a perfect gas inside two-dimensional, axisymmetric, or line-source-type ducts. For the axisymmetric case, provisions to calculate flows with swirl are included. The geometries of the ducts are arbitrary, provided that the flow remains supersonic. The formation of shock waves is automatically predicted, and all shock waves and vortex sheets are treated (using the floating shock fitting technique) as discrete discontinuities that satisfy the appropriate jump conditions. There are no restrictions on the possible number of shock waves and vortex sheets, and all interactions of these discontinuities are evaluated by locally exact solutions. An example of the code applied to the flow field produced by a series of diamond shapes inside a duct (simulating a scramjet combustor) is shown in figure 12 (from ref. 13). Included in the figure are the shock waves (heavy lines), the vortex sheets (dashed lines), and the isobar pattern. Some discontinuities have been eliminated, because they were found to be extremely weak. The effort throughout this work is to provide the designer and the analyst with an accurate, reliable, user-oriented tool.

Exhaust

The afterbody, like the forebody, of an aircraft with an integrated propulsion system must be designed in light of both engine thrust and vehicle aerodynamics. The engine exhaust flow, because of physical area limitations, may be underexpanded at the nozzle exit, and, to obtain maximum propulsive

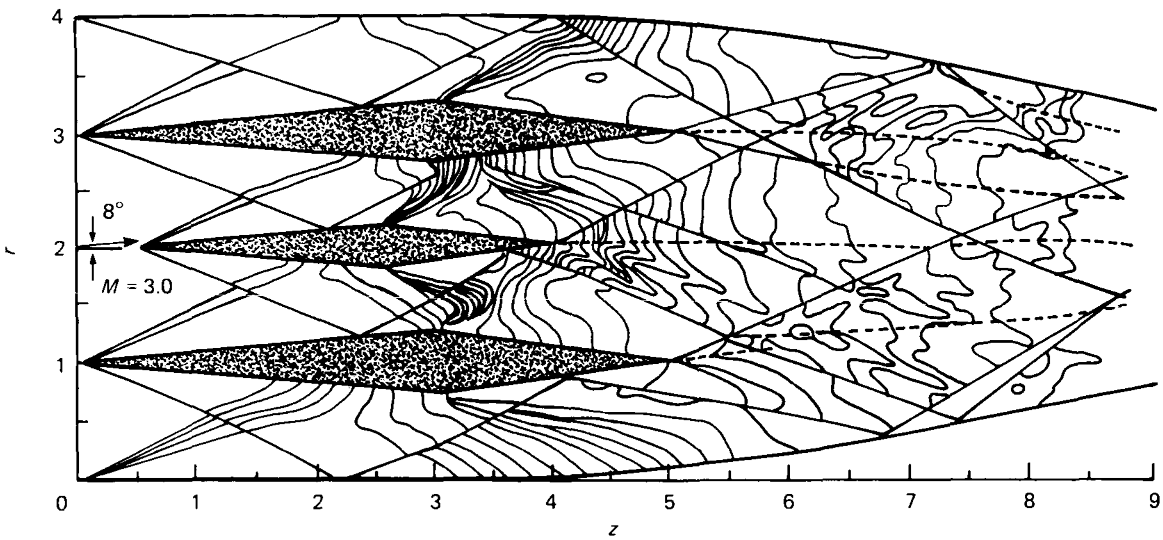


Figure 12.–Simulated scramjet combustor (from Salas, ref. 13).

efficiency, the vehicle afterbody undersurface is used to provide additional expansion. This results in a three-dimensional nozzle flow the boundaries of which are defined both by the solid boundary of the nozzle wall and by the boundary separating the nozzle flow from the vehicle external flow. The dominant features of the scramjet nozzle are as follows:

(1) The gas mixture is composed of hydrogen/air combustion products.

(2) The flow at the combustor exit has nonuniform composition, variable stagnation properties, and a complex wave field.

(3) There is a rapidly expanding internal nozzle flow field with waves generated by and reflected off multiple surfaces.

(4) There is interaction of the nozzle exhaust flow field with the nonuniform vehicle external flow as well as intermodule interactions.

Such flows for supersonic aircraft (see fig. 13, ref. 14) are being studied by Advanced Technology Laboratories, Inc., and NASA Lewis Research Center.

Two programs developed for flow field calculations are CHAR3D and BIGMAC. CHAR3D is a reference plane characteristic code with wave-preserving network, nonisentropic pressure density relation, and conservationlike cross derivatives. BIGMAC is a reference plane finite difference code employing conservation variables and a one-sided

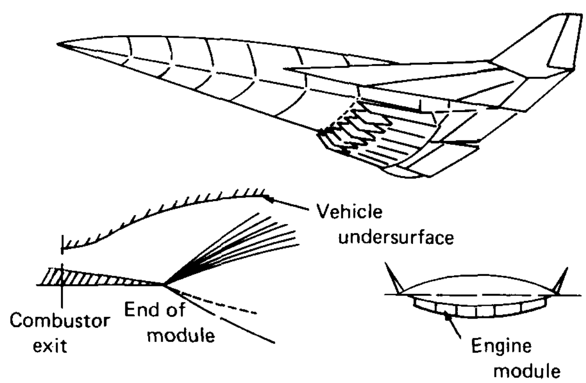


Figure 13.–Exhaust flow field for the hypersonic research airplane (from Dash and Del Guidice, ref. 14).

difference algorithm. Common features of both programs include the following:

(1) Reference plane grid network with respect to three-coordinate systems (Cartesian, cylindrical, and line source)

(2) Quasi-streamlines followed in reference planes

(3) Reference characteristic calculation at all boundary points

(4) Dual cubic spline geometry package

(5) Equilibrium chemistry package

(6) Discrete treatment of plume boundary and underexpansion shock

An example of a calculation for a three-dimensional nozzle exhaust illustrated in figure 14 is shown in

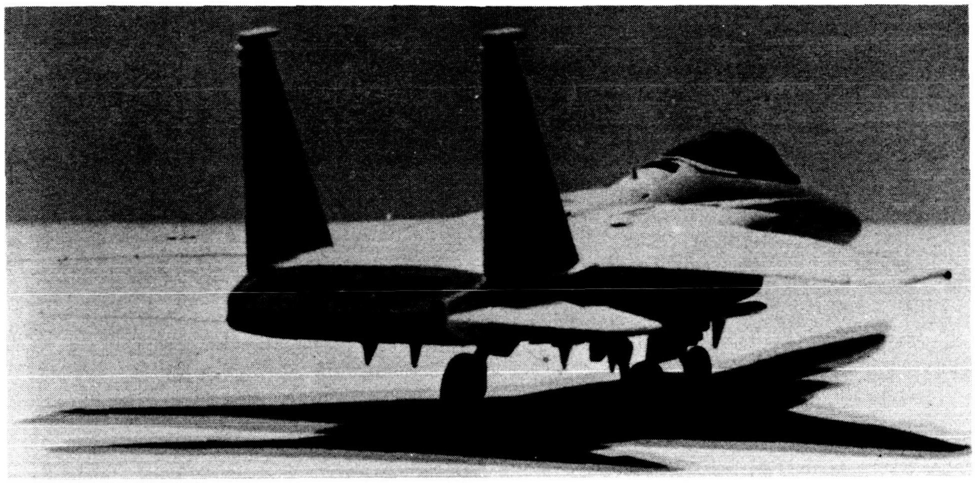


Figure 14.–F-15 with proposed rectangular nozzle.

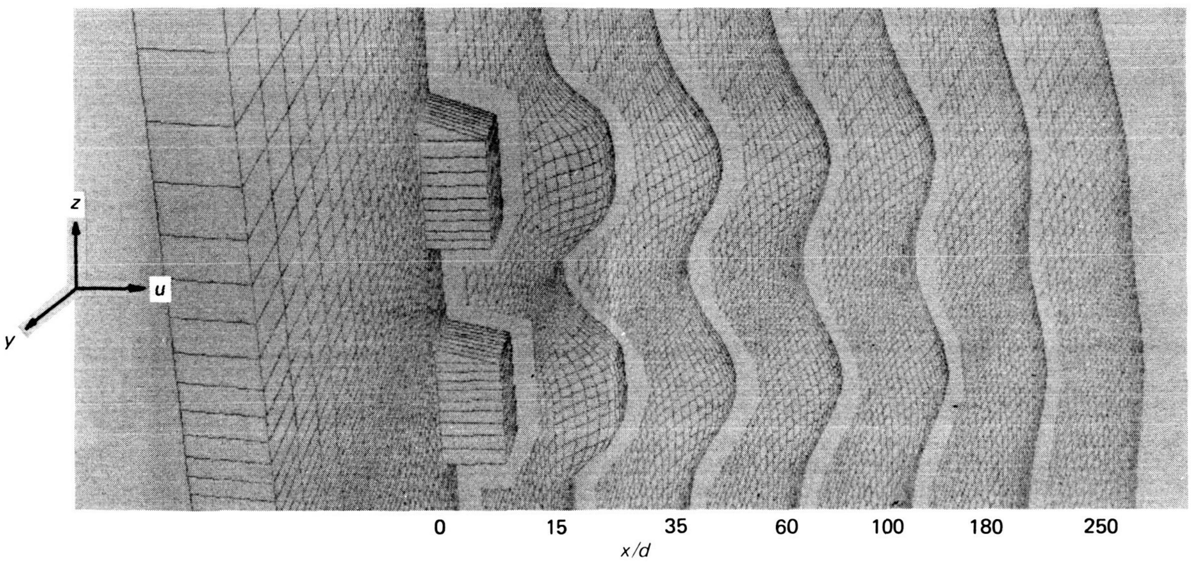


Figure 15.–A simple model of exhaust from a three-dimensional rectangular nozzle.

figure 15. This calculation is the result of an ADI technique applied to the parabolic Navier-Stokes equations.

CONFIGURATIONS

The following discussion on configurations is given in terms of two-dimensional airfoils, subsonic and supersonic aircraft, transonic aircraft, and Space Shuttle applications.

Airfoils

Current computer programs for the computation of two-dimensional, steady, inviscid aerodynamic flows over single element airfoils are fast and flexible, having been highly developed over the past few years. As such they provide a valuable analysis tool for subsonic, transonic, and supersonic flow. A summary of this research is given in table 5.

The approach most widely used to account for viscous effects in an analysis was first suggested by Prandtl (ref. 15). The boundary layer displacement thickness is added to the original geometry and produces an equivalent inviscid shape that represents the displacement of the inviscid flow streamlines by the boundary layer. A number of researchers have coupled an inviscid code with a boundary layer code

TABLE 5.—*2-Dimensional Configurations*

Investigator	Method	Application	Code status
R. L. Barger and C. W. Brooks, Jr.	Finite difference, inviscid	Airfoil design and analysis	Operational; plans include extension to 3 dimensions, rotor blades, and engine inlets
H. L. Morgan	Integral method: coupled boundary layer/potential flow	Subsonic airfoil design and analysis	Operational for analysis of multielement airfoils
E. Murman	Finite difference, inviscid	Subsonic and transonic airfoil design and analysis	Operational; plans include extension to full speed range and coupling with boundary layer code
R. F. Warming and R. M. Beam	Finite difference, hybrid scheme, inviscid	Dynamic airfoil analysis	Operational
E. D. Martin	Semidirect finite difference, inviscid	Transonic thin airfoil analysis	Operational; plans include extension to arbitrary airfoil
R. Hicks	Numerical optimization, inviscid	Subsonic and transonic airfoil design	Operational; plans include developing user package and extension to 3-dimensional wings
G. S. Deiwert	Explicit finite difference, time-marching Navier-Stokes	Transonic airfoil analysis	Operational
P. Bavitz	Coupled boundary layer/ potential flow	Transonic airfoil analysis	Operational
L. Olson	Coupled boundary layer/ potential flow	Multielement airfoil transonic analysis	Operational; plans include extension to 3 dimensions
F. Bauer, P. Garabedian, and D. Korn	Complex characteristics in hodograph plane	Transonic shockless airfoil analysis	Operational; plans include developing design capability
A. Jameson	Finite difference, relaxation, with fast elliptic solver	Transonic airfoil analysis	Operational
L. A. Carlson	Finite difference, inverse	Transonic airfoil design and analysis	Operational
J. E. Carter and S. F. Wornom	Coupled boundary layer/ potential flow	Calculations through small separation regions in airfoil analysis	Operational
T. J. Coakley and J. G. Marvin	Explicit finite difference, Navier-Stokes	Shock-boundary layer interaction	Operational
C. M. Hung	Explicit finite difference, Navier-Stokes	Compression corner	Operational
S. H. Goradia	Integral method: coupled boundary layer/potential flow	Subsonic airfoil	Developmental
U. B. Mehta	Implicit finite difference, Navier-Stokes	Unsteady separation	Operational
J. F. Thompson	Finite difference, Navier-Stokes	Incompressible multielement airfoil	Operational; plans include adding compressibility effects
J. C. Wu	Integro-differential, Navier-Stokes	Unsteady separation on airfoil	Operational

to provide a detailed analysis of the interacting flow field. (These codes have also been used successfully for shock/boundary layer investigations providing the interaction does not cause a large separation bubble to occur.) Results given in figure 16 (from ref. 16) show the complete adequacy of the analysis method

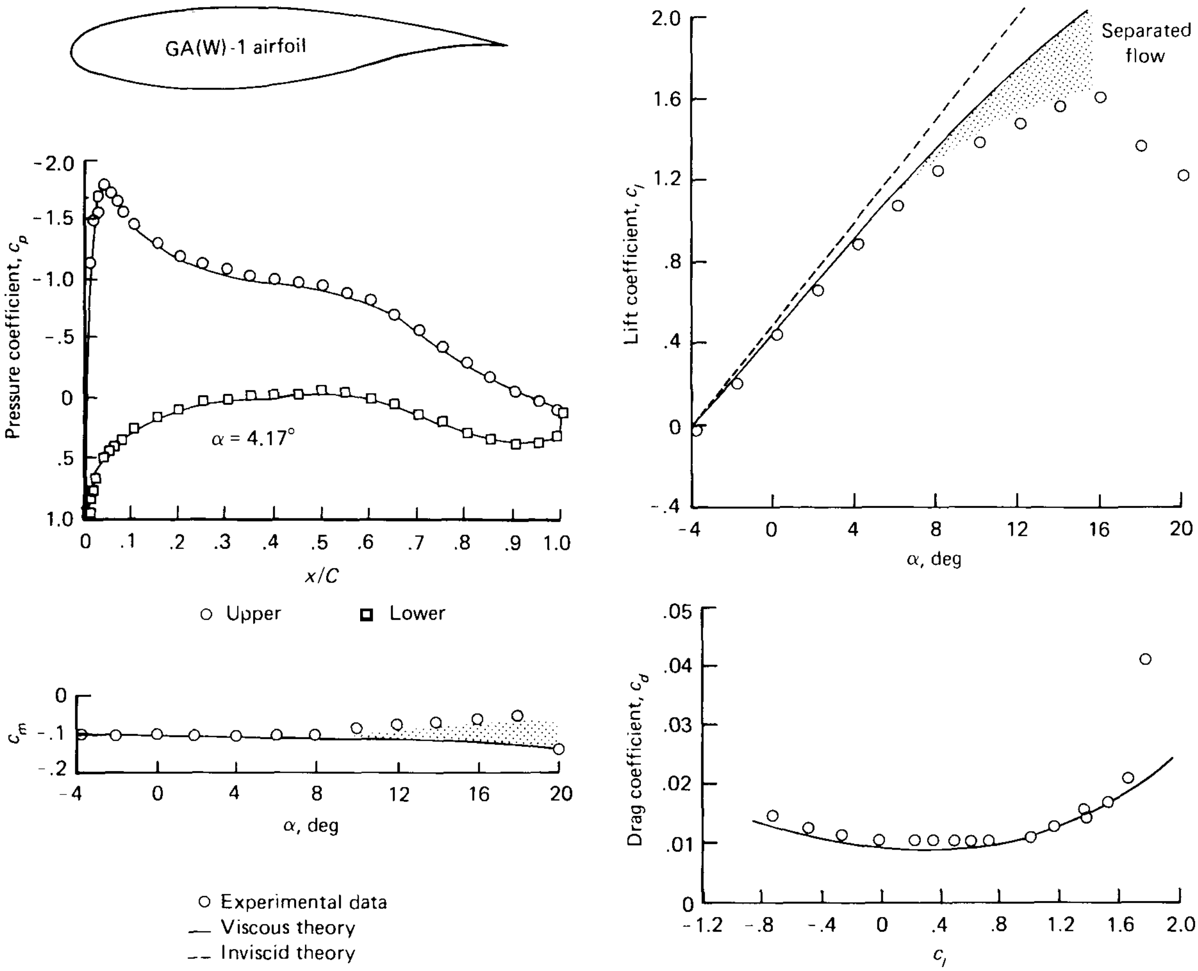


Figure 16.—GA(W)-1 airfoil comparison (from Barger and Brooks, ref. 16). (c_m = mass coefficient.)

for subsonic airfoils at angle of attack below stall. At higher angles of attack, where there is flow separation, the theory breaks down. Viscous theory is sometimes inadequate before separation occurs. Thus, as illustrated in figure 17 (from ref. 17), the theory cannot be depended upon to predict the characteristics of very thick airfoils (e.g., 21 percent) at low Reynolds numbers where large viscous effects dominate.

For analysis of multielement airfoils with high-lift capability, present analysis programs are not as accurate as for single-element airfoils although they are useful engineering analysis and design tools for high-lift devices. The two-element airfoil configuration shown in figure 18 (from ref. 18) and the four-element airfoil configuration shown in figures 19 (from ref. 19) and 20 (from ref. 20) are illustrative of the analysis capability.

Current transonic analysis methods such as those developed at New York University (by Garabedian (ref. 21) and Jameson (ref. 22)) and by Murman (ref. 23) effectively handle the analysis of transonic airfoils. This analysis capability is shown in figure 21 for the case of a supercritical airfoil and in figure 22 for the case of a conventional airfoil (ref. 24). Of particular interest is the good prediction of shock location and strength.

Numerical methods for airfoil design are also highly developed for subsonic and supersonic flow and are in the process of development for transonic flow. The best-known and most widely used design methods are the hodograph methods of Bauer, Garabedian, and Korn (ref. 25). While these methods yield accurate results, they involve complex mappings and transformations. Another design technique is to use a direct method to analyze the flow about a given

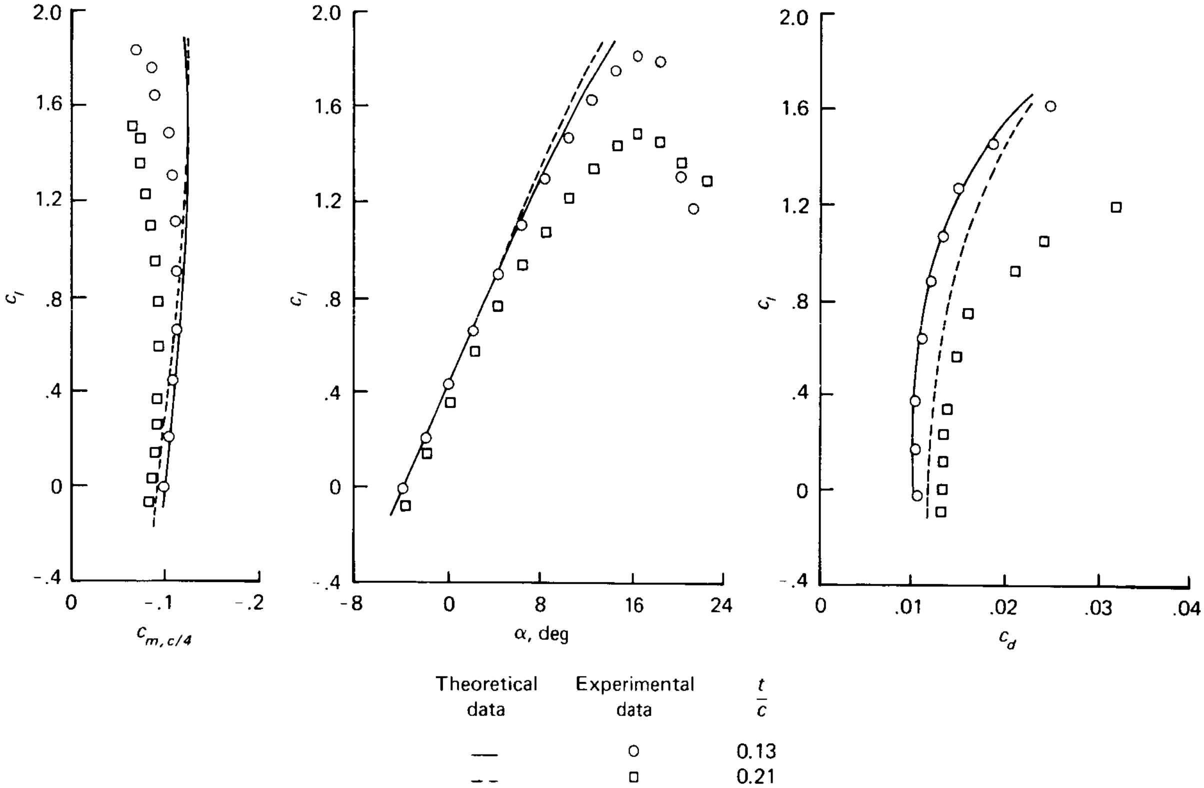


Figure 17.—Analysis of general aviation airfoil (from McGhee and Beasley, ref. 17). $\bar{M}$ = 0.15; R = 3.0 × 10^6; fixed transition.

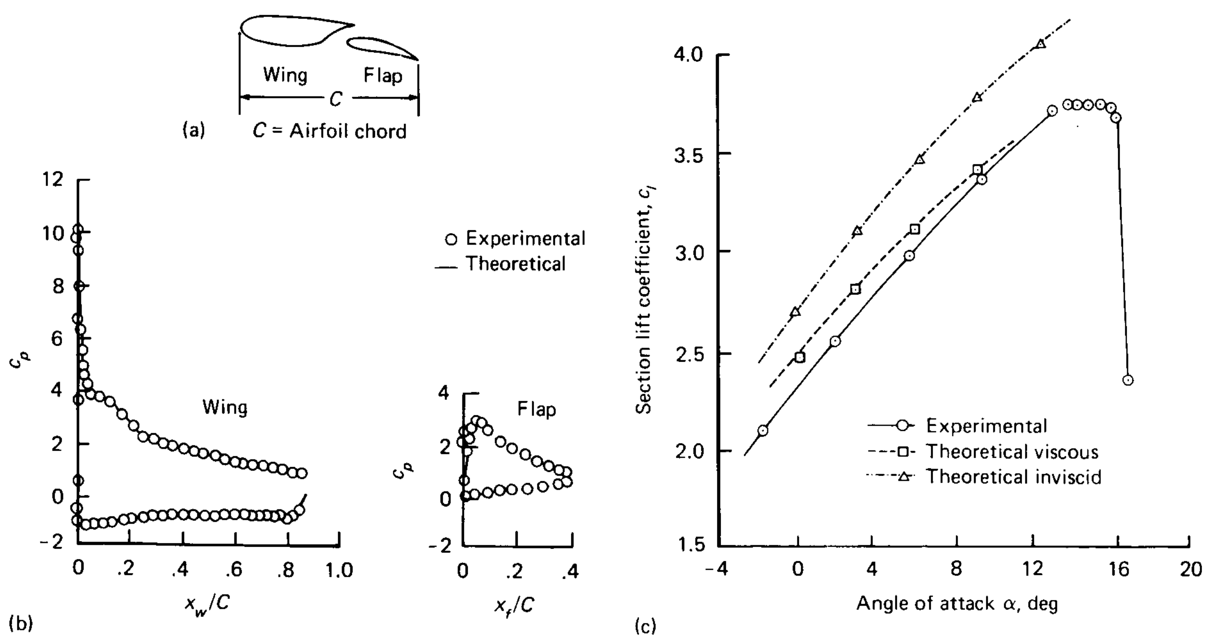


Figure 18.—Analysis of two-element airfoil with single slotted flap (from Olson and Dvorak, ref. 18). δ_{flap} = 30°. (*a*) Diagram. (*b*) Section pressure distribution, α = 7.5°. (*c*) Section lift.

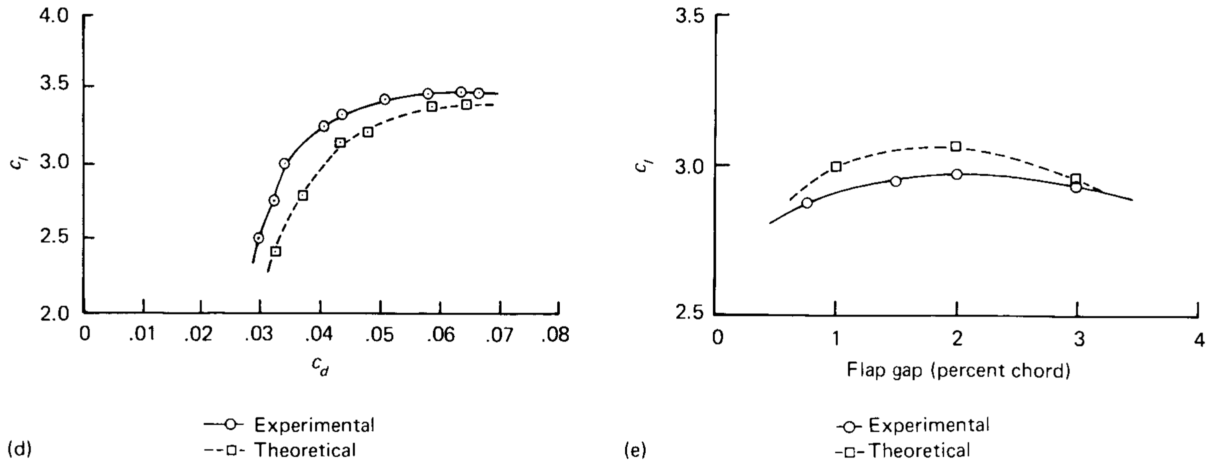


Figure 18 (concluded).–(*d*) Lift versus drag. (*e*) Flap gap optimization.

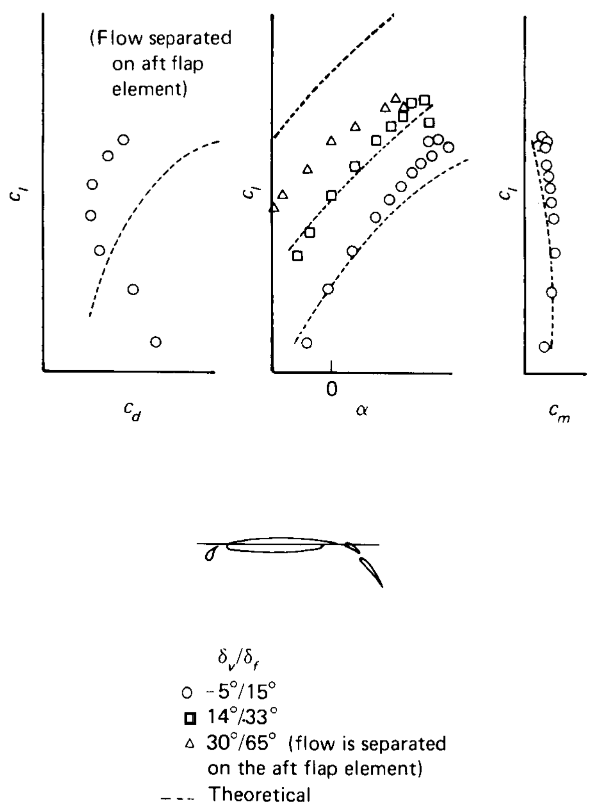


Figure 19.–Analysis of four-element airfoil (from Morgan, ref. 19). $\delta_v = \delta_{vane}$; $\delta_f = \delta_{flap}$; $\delta_{slat} = -45°$.

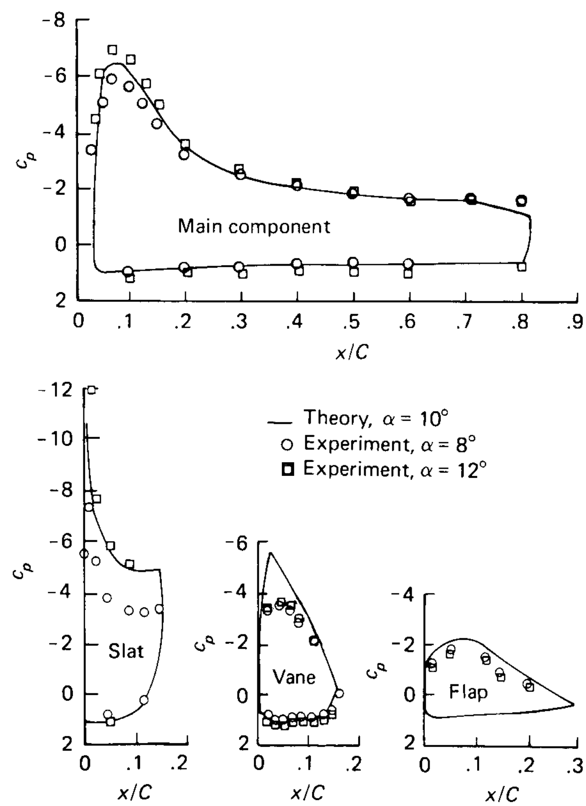


Figure 20.–Analysis of four-element airfoil (from Goradia and Lilley, ref. 20). $\delta_s = -45°$; $\delta_v = 14°$; $\delta_f = 33°$.

airfoil and then, based upon this result, to modify the airfoil to satisfy the design conditions. This usually requires extensive experience on the part of the user and a large number of iterations, but it can give good results. Research is currently underway at NASA Ames Research Center (Hicks, ref. 26) in which a

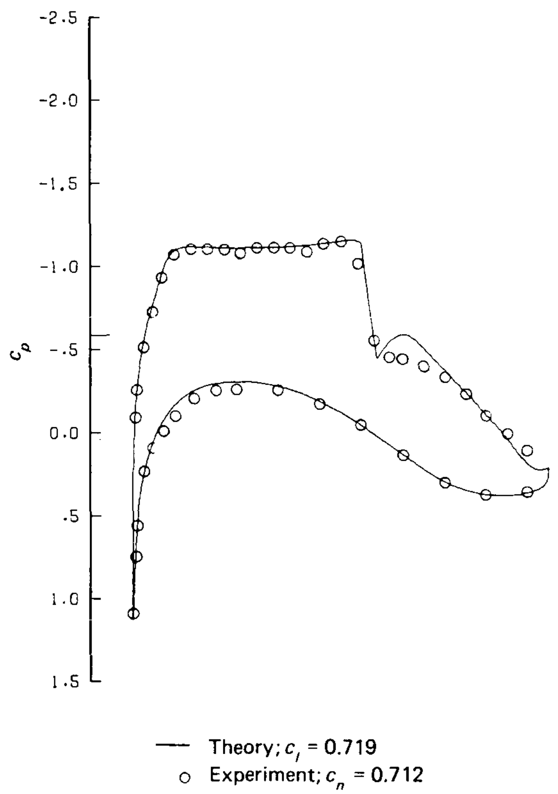


Figure 21.—Analysis of Korn supercritical airfoil (from Bavitz, ref. 24). $M = 0.752$; $R = 20.95 \times 10^6$; $\alpha = 1.25°$.

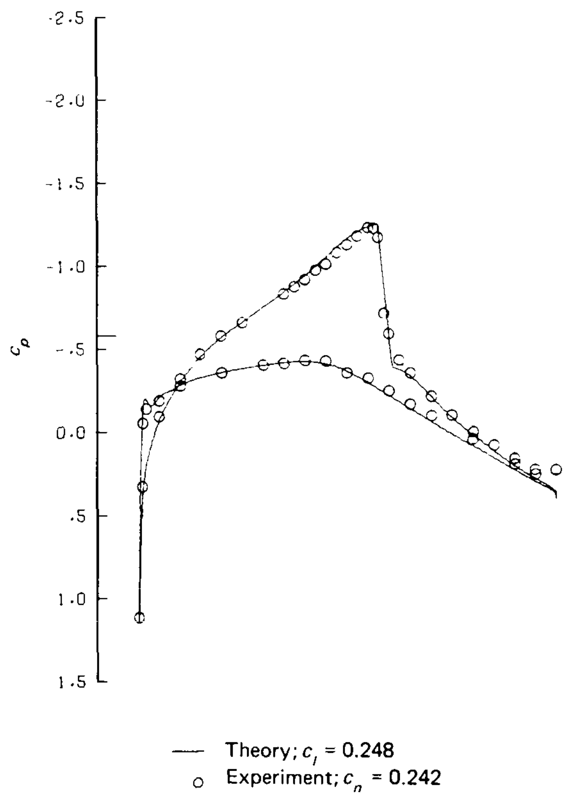
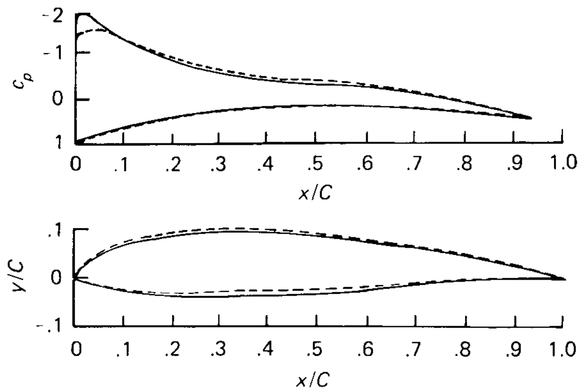


Figure 22.—Analysis of conventional airfoil NACA 65_1-213 (from Bavitz, ref. 24). $a = 0.5$; $M = 0.753$; $R = 52.60 \times 10^6$; $\alpha = -0.10°$.

small perturbation analysis program is combined with an optimization program based on the method of feasible directions. This design capability extends to the optimization of various characteristics, as illustrated in figure 23 (from ref. 27) for supersonic flow and in figure 24 (from ref. 28) for transonic flow. A third design method is an inverse method in which the airfoil surface pressures or velocities are specified and the airfoil shape is determined from the calculation. This approach requires a knowledge of what would be a good pressure distribution. While the user can select a pressure distribution that would result in a given lift and moment and satisfy other flow characteristics, the resultant airfoil design may or may not be physically and/or structurally reasonable. Current research using the inverse method is concerned with providing improved airfoils of practical design.

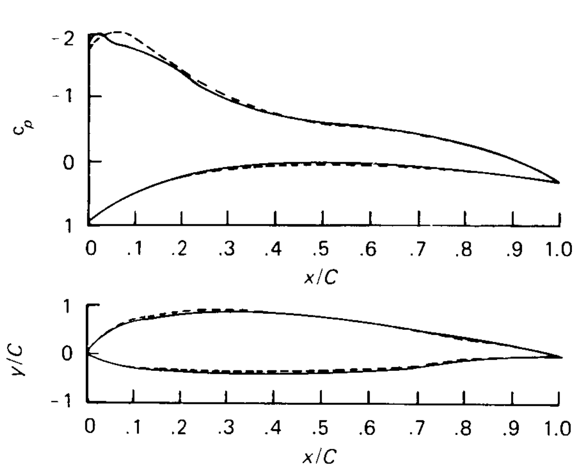
The power of the theory for the analysis and design of airfoils, as opposed to past empirical methods that involved scaling of ordinates to create airfoil families, is evident. However, current theories have been shown to be inadequate for use in the analysis and design of airfoils in situations wherein significant amounts of flow separation occur. Until reliable turbulence modeling procedures or far more powerful computers are developed, it will be necessary to obtain stalled airfoil characteristics by means of careful experimentation. However, several analytical investigations underway show promise of being able to at least qualitatively predict the features of two- and three-dimensional bodies with extensive flow separation. A Navier-Stokes calculation of an airfoil undergoing unsteady laminar separation is shown in figure 25 (from refs. 29 and 30) to reproduce the major flow features revealed in wind tunnel flow visualization studies.

A different approach to flow separation employs a wake description of discrete vortexes arising from the separation of shear layers at the surface. The discrete vortexes connect and diffuse downstream to form an unsteady wake. Figures 26 (from ref. 31) and 27



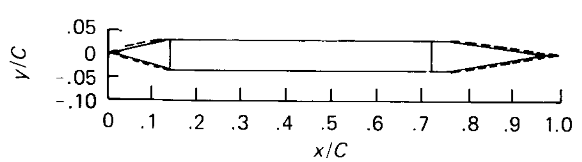
Constraints: $|c_{p_{US}}(x/C = .01)| \leqslant 2.0$ $|c_m| \leqslant .075$ $A \geqslant .075$

		c_l	c_m	A
	— Initial airfoil	0.985	-0.056	0.086
(a)	-- Final	1.085	-.074	.088



Constraints: $|c_{p_{US}}| \leqslant 2.0$ $c_l \geqslant 1.0$ $A \geqslant .075$

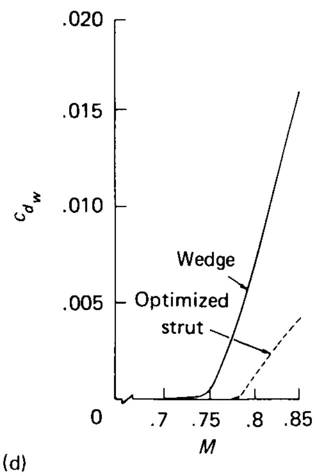
		c_l	c_m	A
	— Initial airfoil	0.9849	-0.0560	0.0863
(b)	-- Final airfoil	1.000	-.0507	.0804



-- Optimum airfoil; $c_{d_w} = .0019$

(c) — Initial airfoil; wedge, $c_{d_w} = .0125$

Figure 23.—Airfoil design by numerical optimization for supersonic flow (from Vanderplaats, Hicks, and Murman, ref. 27). (A = area; US = upper surface.) (a) Lift maximization. $M = 0.1$; $\alpha = 6°$. (b) Pitching moment minimization. $M = 0.1$; $\alpha = 6°$. (c) Wave drag minimization of strut. $M = 0.82$. y/c versus x/c.

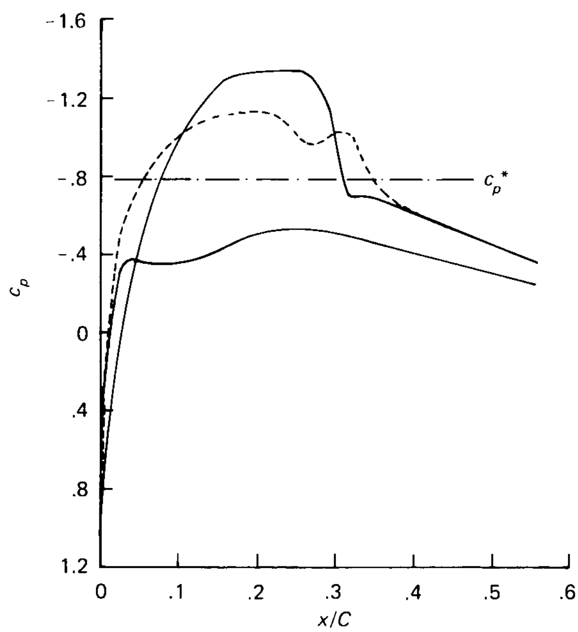


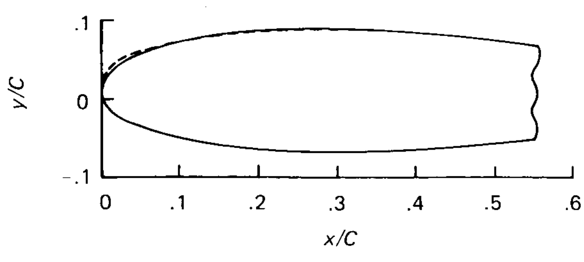
— Initial airfoil; wedge, $c_{d_w} = .0125$

-- Optimum airfoil; $c_{d_w} = .0019$

(d)

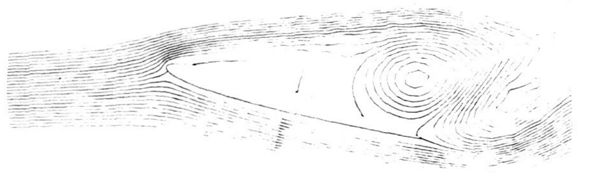
Figure 23 (concluded).—(d) Wave drag minimization of strut. $M = 0.82$. c_{d_w} versus M.





	c_l	c_d	c_m
— NACA 23015	0.235	0.0037	-0.009
-- Modified	.241	.0005	-.007

Figure 24.—Leading edge modification for transonic speeds using numerical optimization (from Jameson, ref. 28). $M = 0.7$; $\alpha = 0°$.


(a)


(b)

Figure 25.–Leading edge stall. (*a*) Angle of attack = 15°; Re = 10^3; 9 percent thick symmetrical airfoil; from Mehta and Lavan (ref. 29). (*b*) Wind tunnel flow visualization (from Prandtl, ref. 30).

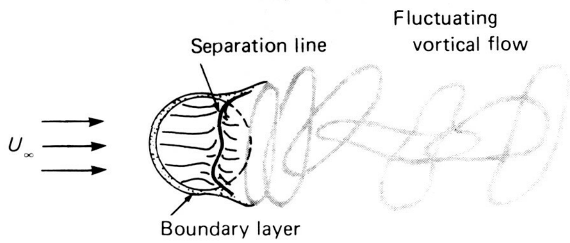


Figure 26.–Schematic drawing of three-dimensional unsteady, separated flow (from Leonard, ref. 31).

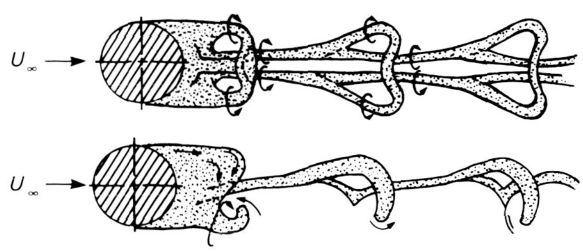


Figure 27.–Vortex structure in the wake of a sphere as observed by Achenbach (ref. 32). Re = 10^3.

(from ref. 32) show the method as applied to a sphere and indicate that the calculations produce vortex flow patterns similar to those observed experimentally. Analyses have been completed for a variety of two-dimensional shapes at angle of attack, and programing is underway to extend the method to arbitrary three-dimensional bodies.

Subsonic and Supersonic Aircraft

At subsonic and moderate supersonic velocities and lower altitudes the airflow around a configuration remains in chemical equilibrium, and design information can be obtained from the perfect gas flow calculations and wind tunnel testing of appropriately scaled models. Current three-dimensional analytical methods for complete configurations are usually based on potential methods and are suitable only for inviscid flow. (See table 6.) The highly developed Boeing program has made extensive use of the paneling procedure for a variety of configurations. For the aerodynamic representation, the configuration surface is divided into a number of panels. A constant source distribution is then imposed on the body panels and a linear vortex distribution is used to represent the wing and tail panels. Typical results for wing pressures for two types of wing/body configurations (a supersonic transport and a maneuvering fighter) are presented in figure 28 (from ref. 33). Current research is directed toward incorporating the subsonic doublet-lattice and vortex spline aerodynamics technology to improve programing accuracy and efficiency.

For high-speed aircraft, the effects of separation-induced vortex flows are becoming increasingly important, particularly optimization of vortex-induced maneuver lift. Prediction of lift and pressure distributions for delta wings at angle of attack have been attempted with linear lifting surface theory, slender wing theory, and the Smith conical method. However, these approaches have been inadequate to handle the vortex flow emanating from the leading edge of such wings. Polhamus' recent leading-edge suction analogy method (ref. 34), which has proven to be extremely successful in predicting the lift on a number of configurations in which vortex flow is a significant contributor, does not predict pressure distributions. A complete three-dimensional paneling solution, which has recently become available from Boeing and which accounts for the free-sheet vortex and frozen wake, accurately predicts such pressures. This free-sheet method is compared with data and some earlier theories in figure 29 (from ref. 35).

Some application of the viscous equations for steady three-dimensional flow is now being made

TABLE 6.—*Subsonic and Supersonic Aircraft*

Investigator	Method	Application	Code status
L. Morino	Finite element	Subsonic and supersonic configuration design	Operational; plans include extension to transonic flow
W. D. Middleton	Linear theory	Integrated system design and optimization	Operational for supersonic flow code currently being optimized
P. Kutler and L. Sakell	Finite difference, inviscid	Shock-on-shock interaction	Operational
J. A. Weber, G. W. Brune, and F. T. Johnson	Paneling	Wing leading edge	Operational for subsonic flow
J. E. Lamar	Suction analogy	Augmented vortex lift interaction effects	Operational; plans include extension to round leading edge wings
P. E. Rubbert, F. T. Johnson, and F. E. Ehlers	Paneling and doublet spline fitting	Full configuration design	Design operational; plans include extensions for dynamic loads and active control configurations
A. Leonard	Coupled boundary layer/ potential flow	Interacting vortex filaments	Operational
E. Lan	Wing plus thick jet theory	Jet exhaust/wing interaction	Operational for subsonic flow with arbitrary jet shape and deflection; plans include extension to higher mach numbers
V. Rossow	Lagrangian marching	Vortex alleviation in incompressible flow	Operational
M. L. Lopez and C. C. Shen	Jet flap theory and elementary vortex distribution	Propulsive lift on wings	Operational; plans include extending calculation to full body
J. R. Tulinius, D. S. Miller, J. L. Thomas, and R. J. Margason	Paneling plus jet flap	Full configuration	Developmental; will be used for configuration design
S. C. Lubard	Fully implicit finite difference, parabolic Navier-Stokes	Analytic configuration in supersonic flow	Operational
S. G. Rubin and C. L. Lin	Two-step finite difference, parabolic Navier-Stokes	Analytic configuration in supersonic flow	Operational
R. S. Hirsh	Finite difference (ADI)	3-dimensional nozzle exhaust	Operational for supersonic flow; plans include extension to subsonic
W. S. Llewellen and R. D. Sullivan	Finite difference, x-marching viscous	Subsonic wake flow (now being applied to vortex mixing)	Operational

where diffusion in one direction can be neglected. The governing equations, the so-called parabolic Navier-Stokes equations, have been applied to analytic configurations in supersonic flow, supersonic three-dimensional nozzle exhausts, and subsonic wake flow, and are currently being applied to subsonic vortex mixing.

The development of computational methods capable of accounting for various jet interaction effects is of interest because of the need to analyze highly integrated propulsion system/airframe concepts. This is a particularly vital area in vertical takeoff and landing (VTOL) research and in the upper-surface blowing (USB) propulsive-lift concept, which is promising for high-performance aircraft with an inherently lower noise level because the engine nozzle and exhaust are located above the wing and thereby shielded by the wing with respect to noises radiated to the ground. Research in this area involves application of wing plus thick jet theory and elementary

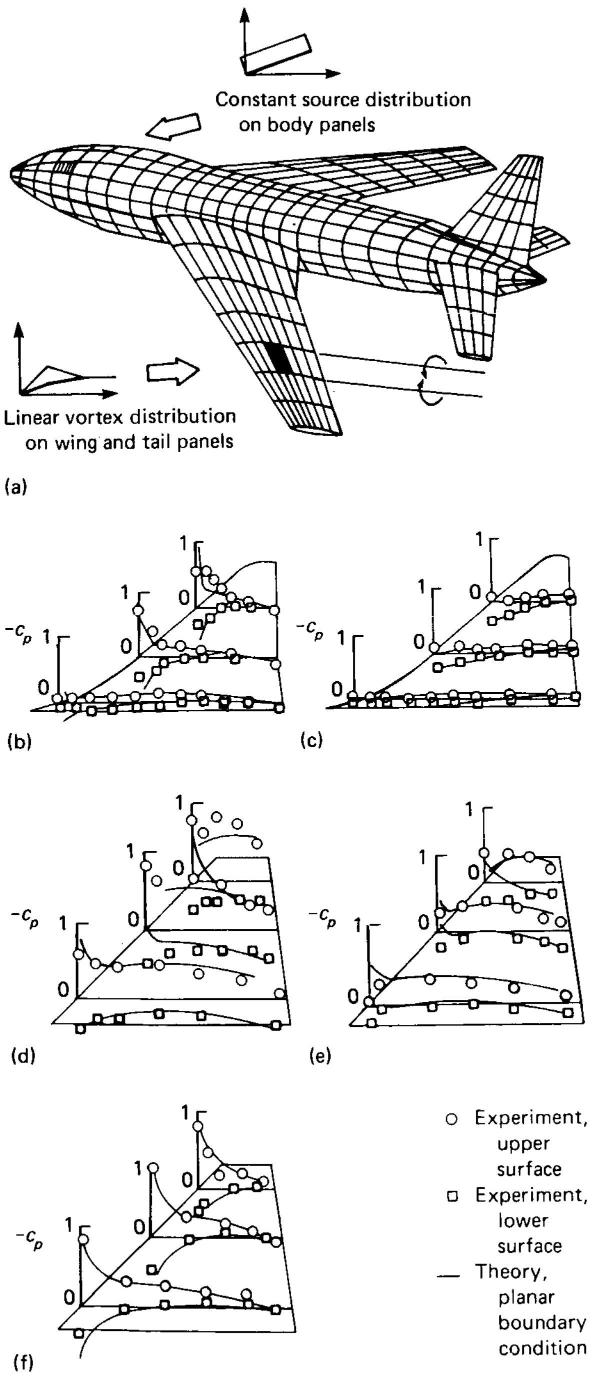


Figure 28.–Pressure calculations from paneling method of Ehlers, Johnson, and Rubbert (ref. 33). (*a*) Aerodynamic representation. (*b*) Supersonic transport wing. $M = 0.60$; $\alpha = 5.63°$. (*c*) Supersonic transport wing. $M = 1.90$; $\alpha = 5.28°$. (*d*) Fighter wing; large camber. $M = 0.40$; $\alpha = 8.37°$. (*e*) Fighter wing; moderate camber. $M = 0.40$; $\alpha = 3.93°$. (*f*) Fighter wing; no camber. $M = 0.26$; $\alpha = 6.4°$.

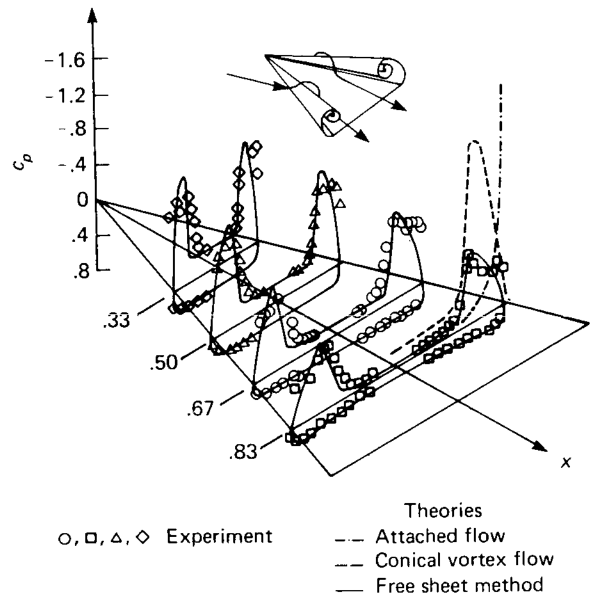


Figure 29.–Edge separation vortex flow (from Weber et al., ref. 35). $A = 1.46$; $\alpha = 14°$; $M = 0$.

vortex distribution. An example of the predictive capability of the thick jet theory in estimating the lift and pitching moment on a USB configuration is shown in figure 30 (from Lan, ref. 36). It can be seen that this theory is a distinct improvement over the thin jet flap approach that neglects the wing/jet interaction effect associated with a thick jet. Dr. Lan is currently extending the theory to account for a round jet blowing over the wing, to examine the relative importance of jet entrainment and inviscid wing/jet interaction effects for configurations at high subsonic speeds, and to account for fighter aircraft designs that have integrated propulsion systems. Unlike the USB concept, a number of propulsive lift approaches involve the use of jet flaps that do not interact with the wing. The elementary vortex distribution (EVD) jet flap theory is shown to be particularly effective in predicting the characteristics of such configurations. An example of such capability is shown in figure 31 (from ref. 38).

Multiple trailing vortices from large aircraft continue to pose a serious hazard for following aircraft. To provide further insight into the rollup process and the mechanisms for eventual breakup of wake vortexes, to verify existing wake vortex predictive techniques, and to supplement experiments, unsteady, three-dimensional numerical simulations of the flow field using vortex filaments are being

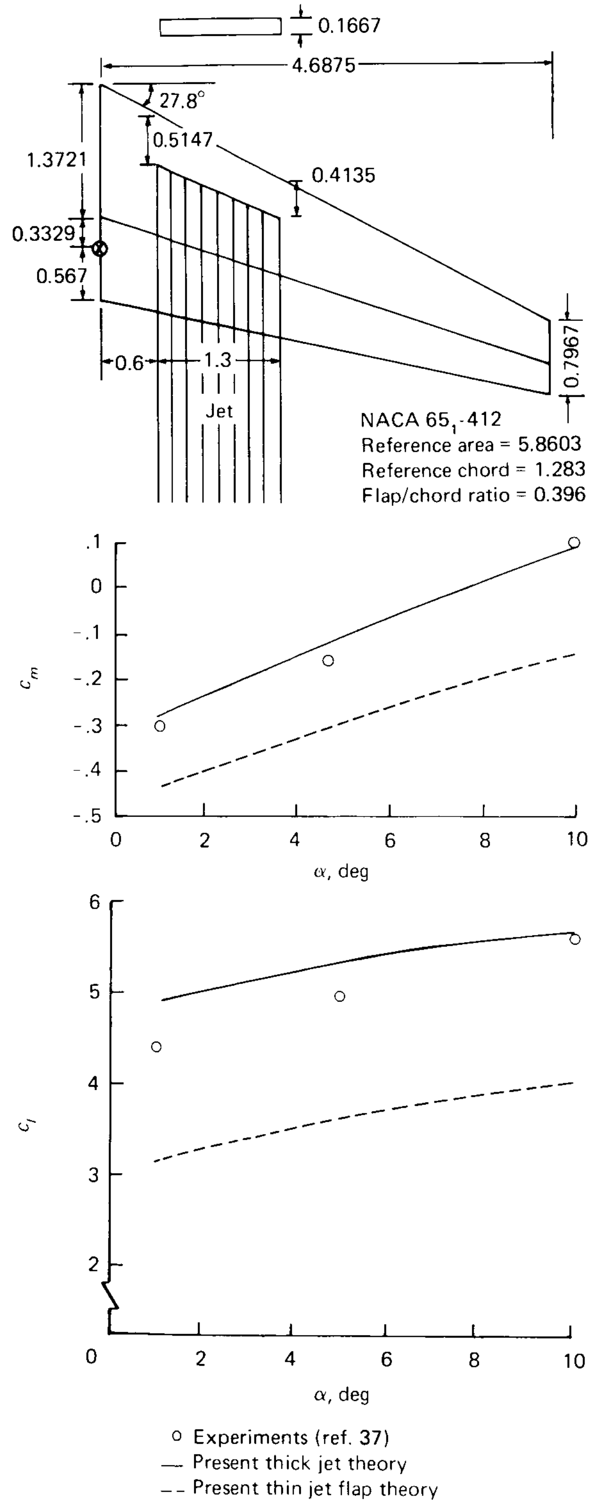


Figure 30.–Jet flap theory for USB configuration (from Lan, ref. 36). (Sketch of USB configuration appears in ref. 37.) The center of gravity is 0.49 below the wing plane.

performed. The multiple vortexes can merge to form a pair of concentrated vortexes with high velocities or can reduce to two diffuse vortexes with low velocities. A study of vortex merging is underway to investigate this phenomenon and analyze the possibility of controlling it. One investigation at NASA Ames (ref. 31) uses a Lagrangian marching procedure. Another simulates the wake vortex flow field by a numerical method based on an Eulerian (fixed grid) finite difference approach (fig. 32). Both the incompressible inviscid and viscous and the compressible inviscid equations for unsteady, two-dimensional flow are being studied as to the relative merits of each for determining the wake flow field. The algorithms for both the compressible and incompressible equations are second-order accurate in time and second- or fourth-order accurate in space. The method used for the incompressible equations is semi-implicit, while the method used for the compressible equations is explicit. After a decision has been made as to which formulation is best, that procedure will be extended to include all three space dimensions. Results of integrating the compressible Euler equations 0.73 span downstream in a wake composed of four vortexes are shown in figure 32(*c*).

Another area where numerical simulations are of interest is the prediction of flow characteristics over missiles and missile-shaped configurations. A method has been developed for solving the exact nonlinear potential equation for axisymmetric flow about blunt or pointed bodies with surface anomalies such as sharp-cornered boattails and flares. The method allows consideration of quite general shapes and is operational throughout the speed range. In the results shown in figure 33 (from ref. 39) for a subsonic freestream, the flow becomes supercritical in the shoulder region, shocks down on the cylindrical portion, expands again to supersonic at the sharp boattail convex corner, and is shocked down again. The calculated flow compresses again to nearly stagnation pressure in the concave corner, which is obviously a poor simulation of the wake separation. Figure 34 (from ref. 39) shows results of a calculation for a typical launch vehicle shape in supersonic flow with a simulated exhaust plume.

Transonic Aircraft

Most modern high-speed aircraft achieve optimum cruise performance and maneuverability when flying in the transonic regime where the flow is extremely

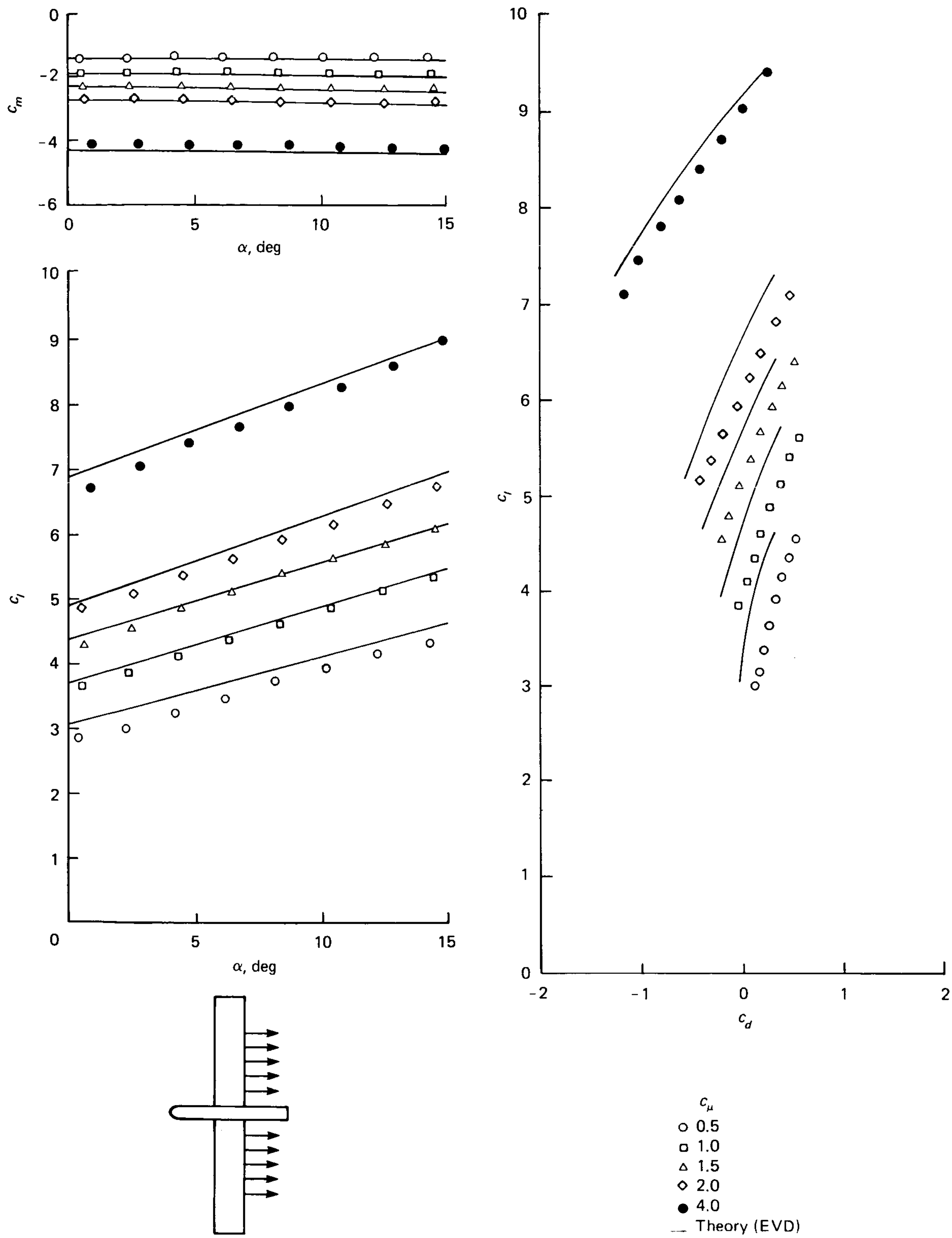


Figure 31.–Jet flap theory (from Lopez and Shen, ref. 38).

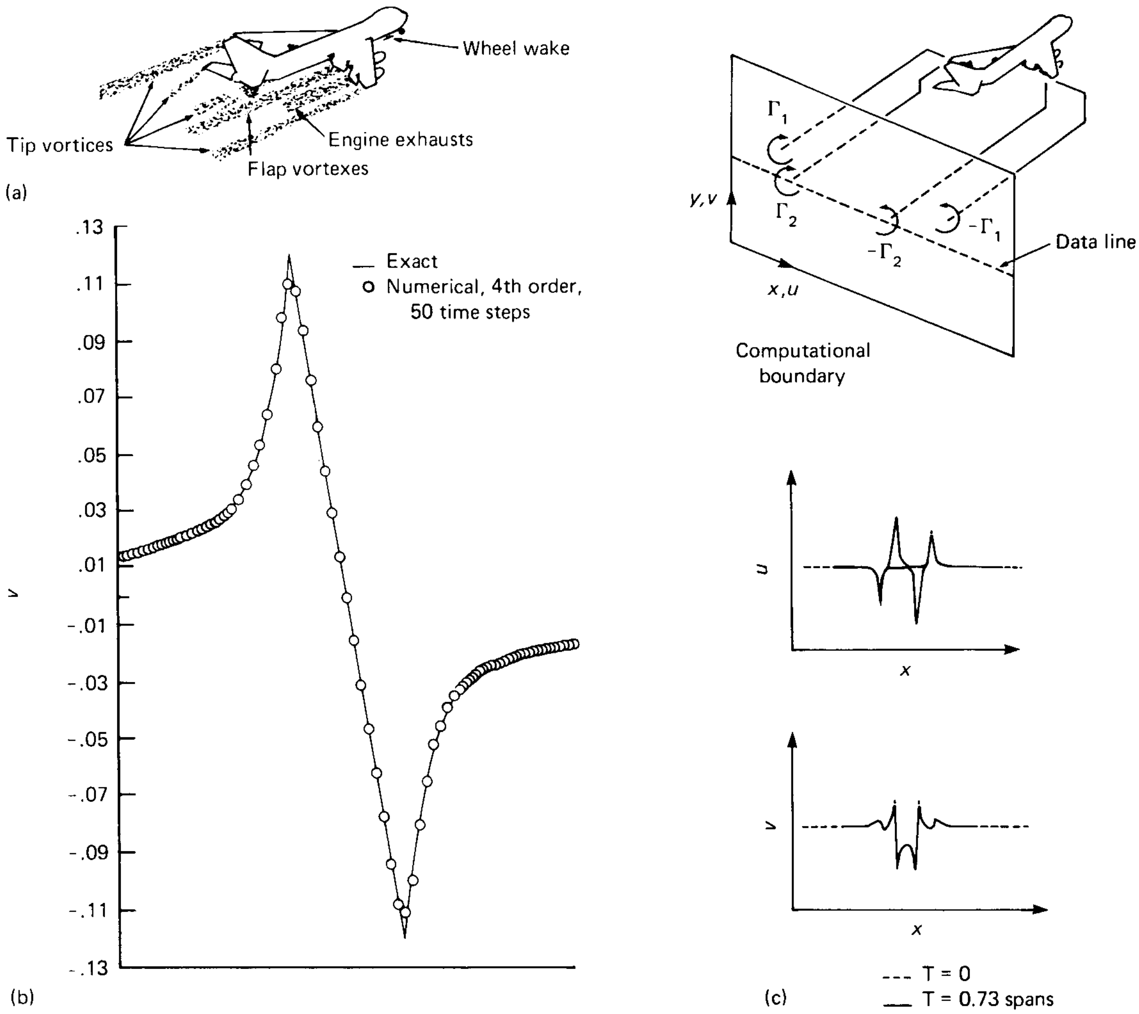


Figure 32.–Wake vortex problem (from Paul Kutler, unpublished). (*a*) Diagram. (*b*) Accuracy check for incompressible, inviscid vortex. (*c*) Multiple vortex pattern (compressible equations).

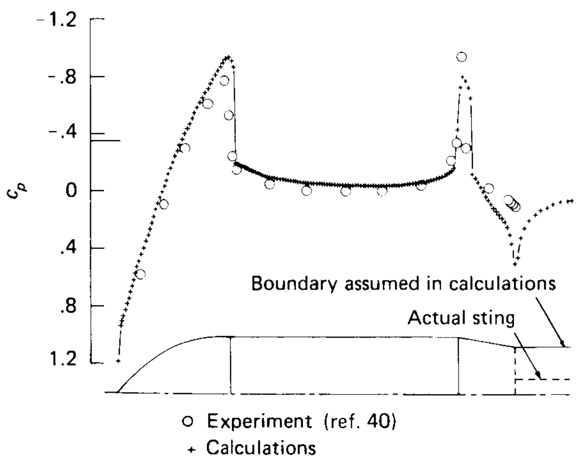


Figure 33.–Pressure distribution for ogive/cylinder/boattail (from Keller and South, ref. 39). M = 0.83; 193 X 49 mesh.

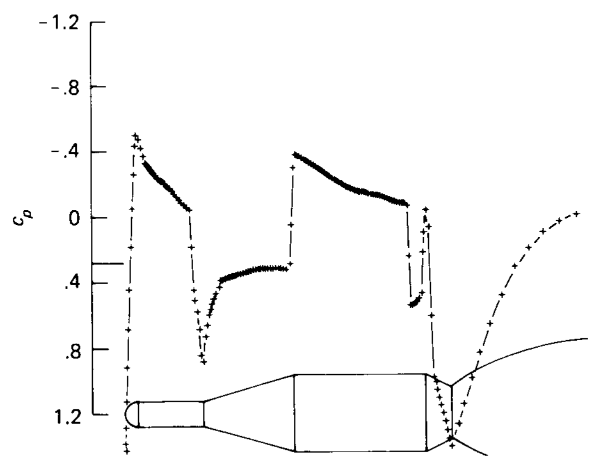


Figure 34.–Pressure distribution for staged vehicle with simulated plume (from Keller and South, ref. 39). M = 1.2; 193 X 49 mesh.

TABLE 7.—*Transonic Aircraft*

Investigator	Method	Application	Code status
P. A. Newman and E. B. Klunker	Finite difference, inviscid, small disturbance approximation	Full configuration	Operational
F. R. Bailey and W. F. Ballhaus	Finite difference, relaxation, inviscid, small disturbance approximation	Wing/body interaction	Operational
J. C. South, Jr., and J. D. Keller	Finite difference, relaxation, mapped coordinates, full potential	Missile shapes	Operational
R. W. Barnwell	Finite difference, relaxation	Wing/body interaction	Operational
P. R. Garabedian and D. Korn	Complex characteristics, full potential	Wings	Operational
A. Jameson	Finite difference, relaxation	Swept and yawed wing	Operational

sensitive to aircraft configuration. Because of the special problems associated with both cruising and maneuvering flight at near sonic speed, a significant effort is being made to develop theoretical transonic flow analysis methods. (See table 7.) The highly nonlinear nature of the equations makes linear analysis of little value; consequently, solutions to the nonlinear governing equations must usually be obtained.

The status of the present effort aimed at computing transonic flows over wing/body configurations by solving finite difference approximations to the transonic small-disturbance equation is illustrated in figures 35 to 39 (from refs. 41 and 42). The conservative finite-difference equation is derived by applying the divergence theorem to the conservation integral of the governing equation over a computational cell. Such an approach assures that the shock jump relations implied by the governing equation are obtained in the finite-difference approximation. Comparisons of computed and experimental pressure distributions for the C-141 wing shown in figure 35 illustrate the accuracy of the technique in predicting the shock strength and location. (The comparison also shows that, in this case at least, inviscid calculations predicted the flight situation more accurately than low Reynolds number wind tunnel tests.) Results are shown in figures 36 through 38 for several isolated wings and in figure 39 for a wing/body configuration. In these figures, the two types of computed results are fully conservative relaxation (FCR), which models the true inviscid shock jump,

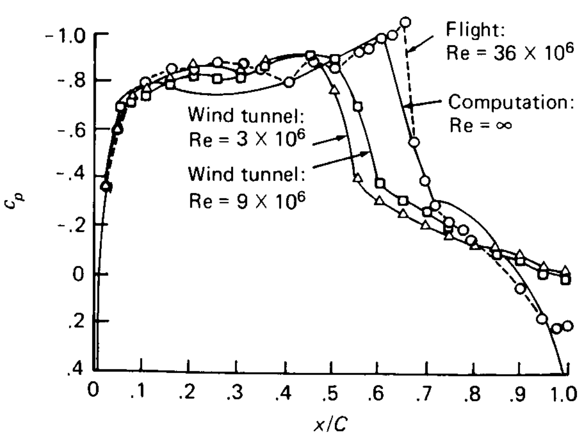


Figure 35.—Comparison of flight, wind tunnel, and computed pressure distribution on upper surface of C-141 wing from Lomax, Bailey, and Ballhaus (ref. 41). $M_\infty = 0.825$.

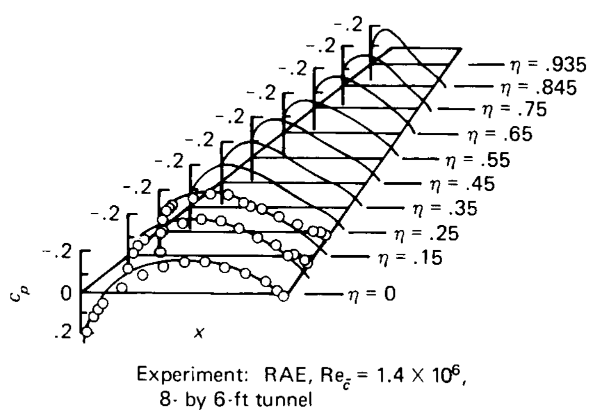


Figure 36.—Pressure distribution for Royal Aircraft Establishment (RAE) wing C (from Bailey and Ballhaus, ref. 42). $M_\infty = 0.95$; $\alpha = 0°$; sweep angle $\Lambda_{C/2} = 44°$; aspect ratio (AR) = 3.6; taper ratio (TR) = 0.333; RAE 101 section; 5.4 percent streamwise. (FCR = fully conservative relaxation; NCR = nonconservative relaxation; Re_c = Reynolds number based on chord.)

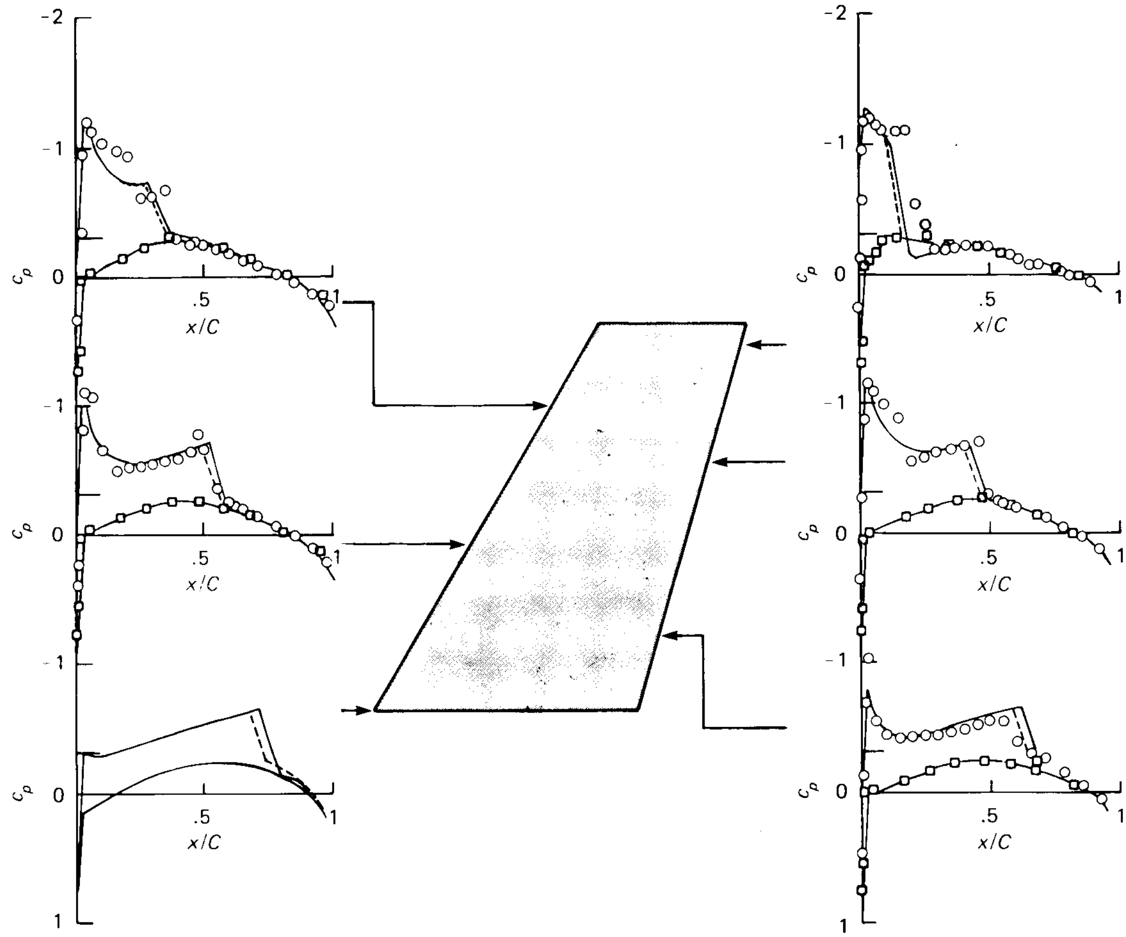


Figure 37.—Pressure distribution for the ONERA M6 wing (from Bailey and Ballhaus, ref. 42). M_∞ = 0.84; α = 3.0°; Λ_{LE} = 30°; AR = 3.86; TR = 5.83; ONERA D section; 9.8 percent streamwise.

and nonconservative relaxation (NCR), which obtains a modified shock jump that often agrees better with experimental results because of viscous effects.

Tunnel wall interference plays an important part in transonic flow experiments; therefore, inclusion of tunnel wall boundary conditions is necessary in all transonic calculation codes before valid comparisons with experimental data can be made. Calculations to simulate a number of conventional tunnel wall boundary conditions have recently been made using a relaxation technique to solve a small disturbance potential equation. Initial numerical results for finite lifting wings show that the embedded supersonic bubble, generally terminated by a shock wave, is distorted compared with that for free air. Computed tunnel wall pressure distributions for several conventional linear boundary conditions compared with the free air results at the wall location are shown in figure 40 (from ref. 43). The present nonlinear finite difference formulation of the problem affords far more flexibility with respect to the tunnel wall boundary condition than was possible in conventional linear correction theory.

For some applications, however, approximate methods are satisfactory and substantial reductions in computer time and storage can be realized. For these

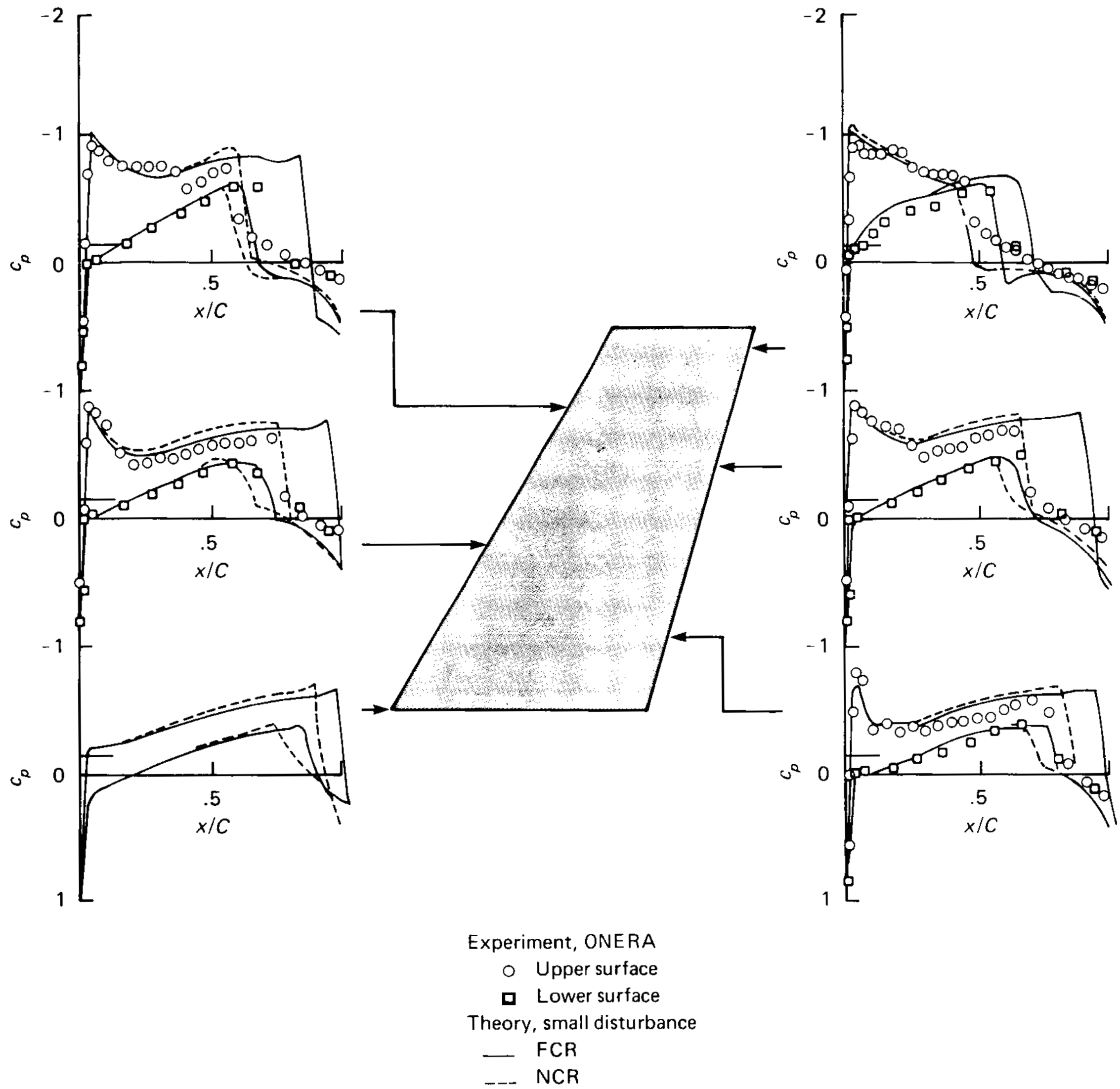


Figure 38.–Pressure distribution for the ONERA M6 wing (from Bailey and Ballhaus, ref. 42). M_∞ = 0.92; α = 3.0°; Λ_{LE} = 30°; AR = 3.86; TR = 0.538; ONERA D section; 9.8 percent streamwise.

applications a rapid approximate method for calculating transonic flow about lifting configurations was developed through an analytical study of the effect of lift on the transonic area rule. This method reduces the three-dimensional fluid mechanics problem to a two-variable computational problem. A highly efficient finite difference relaxation procedure is then applied. Calculations using this method to illustrate the effect of including tunnel wall boundary conditions are shown in figure 41 (from ref. 44). This wall interference effect is observed as a shift in the shock wave and sonic line locations.

The examples presented in this section on transonic aircraft have been computed solutions to the small disturbance equation, except for the approximate method based on the lifting area rule. Some solutions to the full-potential equation (exact compressible inviscid equation) now exist for simple geometric configurations such as missile shapes and moderately swept wings. An example of the analysis capability of solving the full potential equation for a relaxation technique for transonic flow calculation was presented in figures 33 and 34. Extending transonic flow calculations to include viscous effects, either through coupling an inviscid procedure with a boundary layer procedure or by solving the Navier-

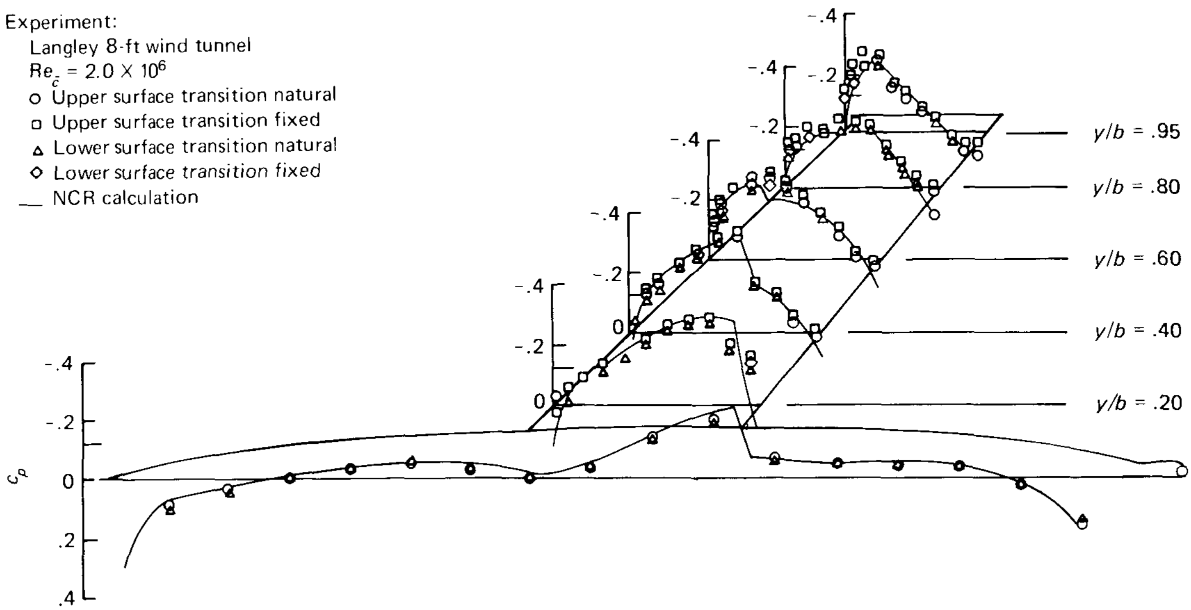


Figure 39.—Pressure distribution for swept wing/fuselage configuration (from Bailey and Ballhaus, ref. 42). $M_\infty = 0.93$; $\alpha = 0°$; $\Lambda_{C/4} = 45°$; AR = 4; TR = 6; NACA 65A006 streamwise.

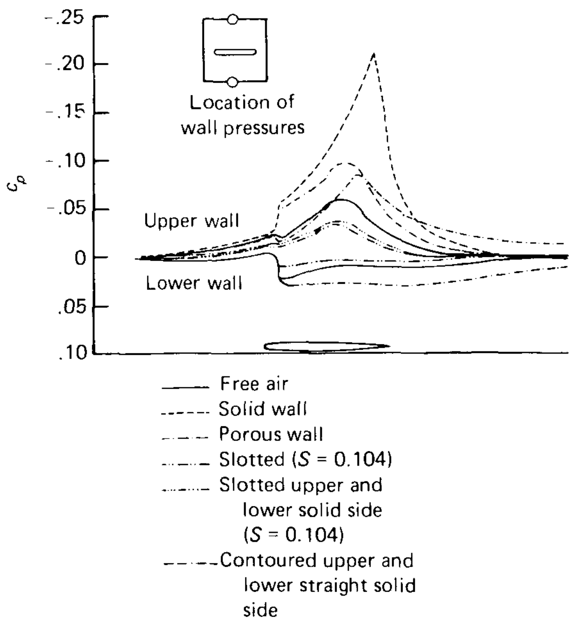


Figure 40.—Tunnel wall pressure distributions in the wing root plane (from Newman and Klunker, ref. 43). NACA 63A006 section; rectangular plan; AR = 32/9; $M = 0.93$; $\alpha = 2.76°$; $h/c = 5.3$.

Stokes equations for transonic flow over configurations, is not feasible without a more detailed understanding of the transonic flow regime. Both detailed experimental data and numerical results including tunnel wall effects are required to determine a satisfactory means of approximating the effects of complex boundaries.

Space Shuttle

At high velocities and altitudes, significant nonequilibrium effects will exist over a large area of the Space Shuttle vehicle. The calculation of flow with finite rate chemistry is, therefore, needed by the Space Shuttle designer as a source of information relative to the prediction of heat transfer rates, boundary layer effects, and aerodynamic loads acting on the aircraft. Complex chemically reacting flow phenomena cannot be scaled, and because current test facilities are incapable of performing full-scale experiments in this regime, the designer must turn to the computer solution for a realistic description of the flow field at actual flight conditions. The majority of the present calculations for the Space Shuttle involve the solution of the inviscid equations. For previous manned atmospheric entry vehicles, nonequilibrium effects could be determined by boundary layer analysis alone, using equilibrium properties for the inviscid external flow. For the Space Shuttle, however, a large part of the inviscid flow will also be nonequilibrium; therefore, a fully reacting inviscid

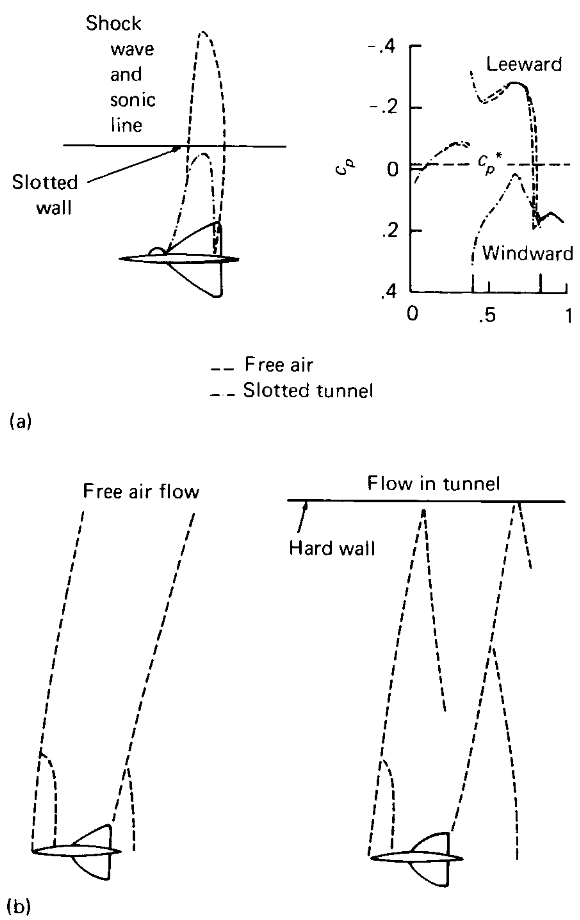


Figure 41.–Tunnel wall effects in transonic flow (from Barnwell, ref. 44). (*a*) Wing plane results for free air and slotted tunnel. $M_\infty = 0.98$; $\alpha = 12°$. (*b*) Effect of tunnel wall on shock wave and sonic line locations. $M_\infty = 1.02$; $\alpha = 10°$.

solution is needed to provide the necessary boundary conditions for a solution of the reacting boundary layer.

Blunt body flows develop a thin entropy layer near the body surface in which strong gradients of flow properties occur. This entropy layer occurs also in reacting flows but is more complex because of the chemical species gradients. Bow shock shapes calculated using a code based on the method of characteristics are shown for the side and plan views and for two cross sections in figure 42 (from ref. 45). To provide some feel for the effects of nonequilibrium chemistry, the Space Shuttle results computed incorporating real gas effects are compared with perfect gas (frozen) computations at the same free stream

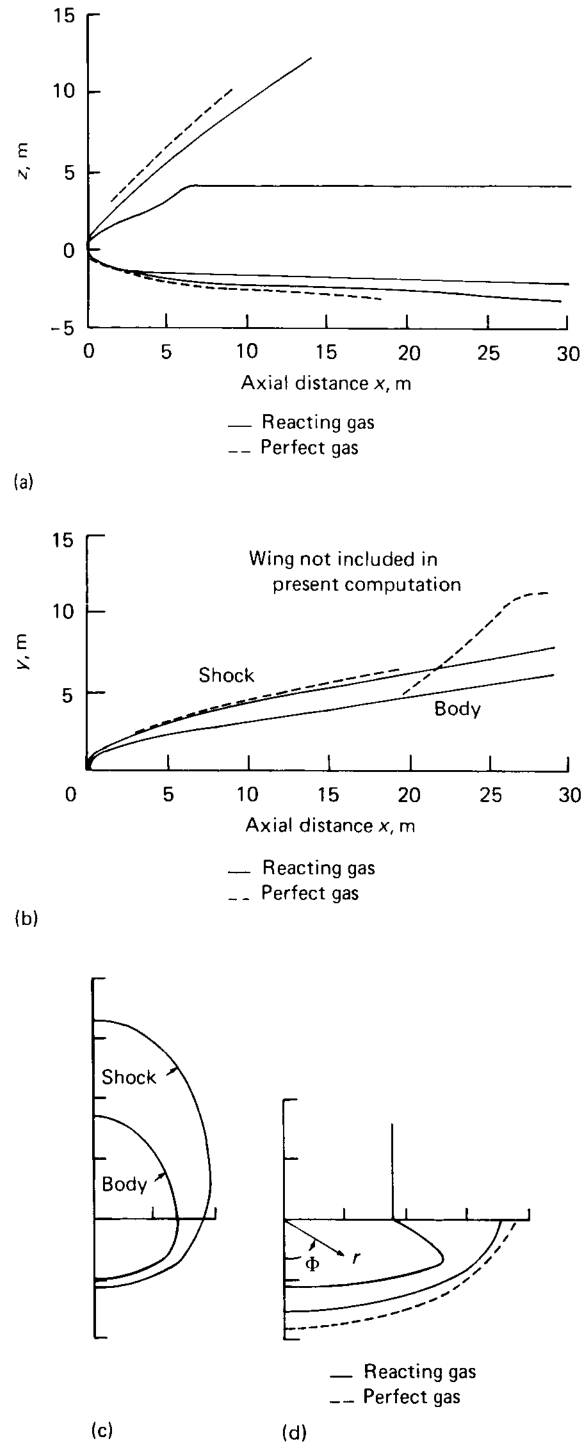


Figure 42.–Shock shape for Space Shuttle 147 body (from Rakich and Kutler, ref. 45). $\alpha = 30°$; $V_\infty = 6.7$ km/s; altitude = 65.5 km. (*a*) Side view. (*b*) Plan view. (*c*) Cross section at $x = 4.75$ m. (*d*) Cross section at $x = 15.2$ m.

mach number. The shock standoff distance is decreased by the chemical reactions. Note that the location of the point of bow shock impingement on the wing is important to the heat shield design.

At moderate angles of attack (up to approximately 30°), the subsonic portion of the flow over winged lifting vehicles, such as the Space Shuttle, is confined to the nose region. An effort at Grumman has resulted in the development of an inviscid computer code for solution of the moderate angle of attack problem (30° to 35°) that provides inputs for a three-dimensional boundary layer code. All shock waves are computed explicitly as discontinuities in the flow. Figure 43 (from ref. 46) shows a comparison of calculated and experimental pressure distributions along the windward and leeward symmetry plane of a Space Shuttle configuration at angle of attack. The agreement is generally good even on the leeward side, except in the nose region where the difference is thought to be due to an oversimplification of the vehicle geometry for the calculations. More accurate geometric modeling will solve this problem.

Calculations for the Space Shuttle configuration using both a three-dimensional inviscid code coupled with a boundary layer code and a three-dimensional Navier-Stokes code were made. Computed and experimental heating rates are compared in figure 44 (from ref. 47) for the windward and leeward centerlines and for one transverse section. As illustrated, the three-dimensional Navier-Stokes solution is extremely "user sensitive" in regard to finite-difference grid selection,

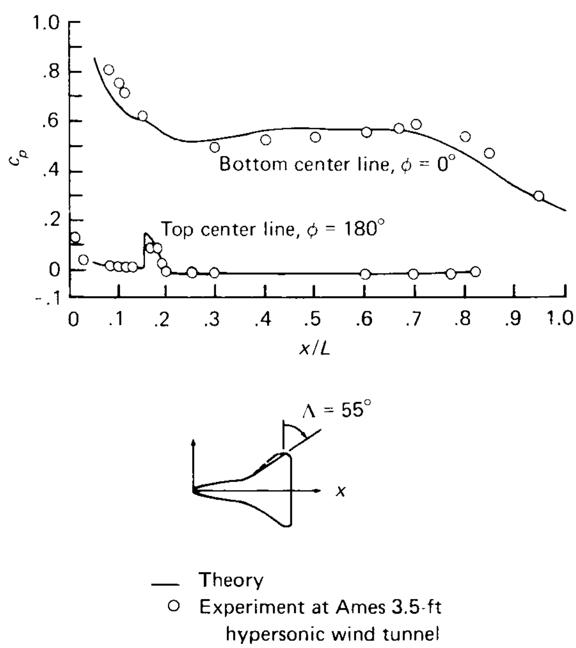


Figure 43.–Longitudinal surface pressures in the Space Shuttle body (from Marconi, Yaeger, and Hamilton, ref. 46). M = 7.3; γ = 1.4; α = 30°; L = 107.8 ft.

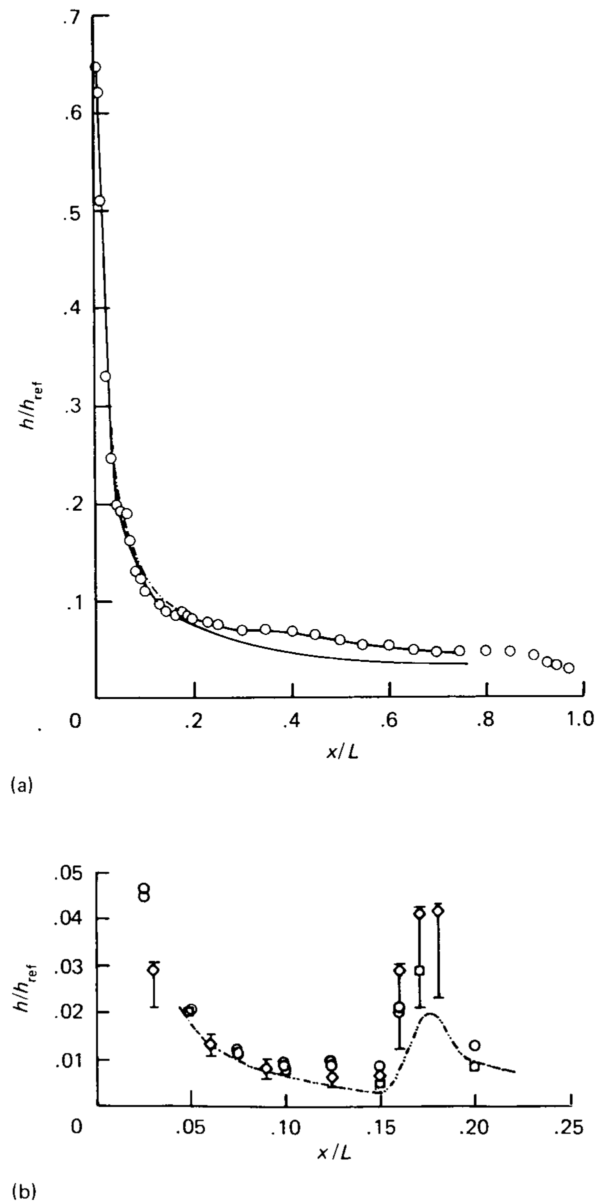


Figure 44.–Heating rate on Space Shuttle (from Goodrich et al., ref. 47). (*a*) Windward centerline. (*b*) Leeward centerline. α = 30°. (See key on next page.)

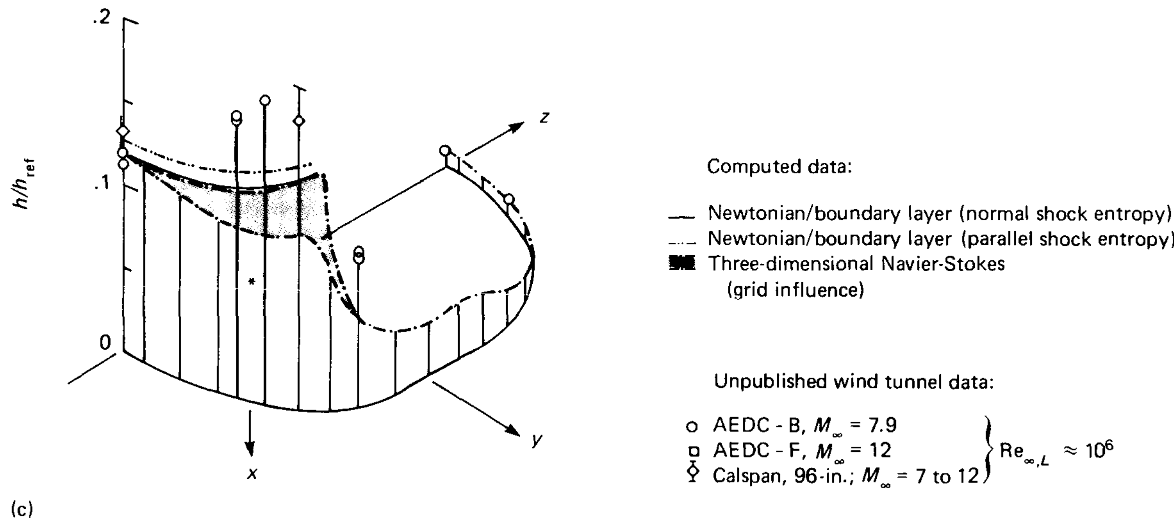


Figure 44 (concluded).–(*c*) Transverse section, x/L = 0.1.

TABLE 8.–*Space Shuttle*

Investigator	Method	Application	Code status
A. W. Rizzi	Time-split finite difference, inviscid	Nonequilibrium flow, chemically reacting	Operational
J. V. Rakich and E. B. Pegot	Coupled boundary layer, inviscid	Windward side heating prediction, laminar flow, perfect gas	Operational
C. P. Li, C. K. Houston, and R. M. Meyers	Coupled boundary layer/ potential flow, finite difference, Navier-Stokes	Full configuration with real-gas effects	Operational
F. Marconi and L. Yaeger	Finite difference, inviscid	Full configuration with real-gas effects	Operational
P. Kutler	Finite difference, inviscid	Unsteady shock interaction	Operational

a feature that discourages its use as a design tool at the present time. There is currently a need for more physical and numerical analyses in addition to more computer power to aid in resolving the difficulties of three-dimensional Navier-Stokes calculations. (See table 8 for a summary of this research in this area.)

CONCLUDING REMARKS

As illustrated in table 9 and figures 45 and 46, the cost of numerical simulation continues to decrease as computer speed and storage continue to increase; consequently, simulations of practical problems of interest to the aerospace community are becoming a reality. Computer codes to calculate solutions of the inviscid approximations and turbulent boundary layer approximations to the Navier-Stokes equations are highly developed and are currently being applied in aircraft design. Current research is directed toward developing numerical procedures, turbulence transport models, and computer codes for obtaining solutions to the time-averaged Navier-Stokes equations.

TABLE 9.—*Arithmetic Instruction Timings in Seconds*

Computer	Operation		
	+/-	×	÷
BABBAGE (1800)	1	60	60
MARK 1 (1937)	.3	6	11.4
ENIAC (1946)	2×10^{-4}	3×10^{-3}	6×10^{-3}
CDC 6600 (1963)	4×10^{-7}	5×10^{-7}	2×10^{-6}
ILLIAC IV	6×10^{-9}	7.8×10^{-9}	4.9×10^{-8}
STAR 100	2×10^{-8}	4×10^{-8}	4×10^{-8}

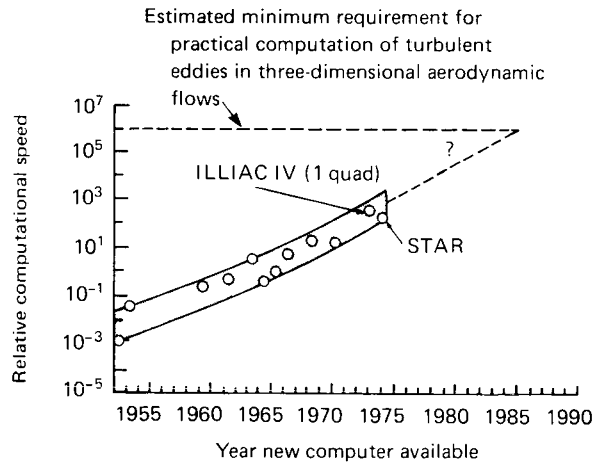


Figure 46.—Trends of computer speed.

Numerical solutions of the time-dependent Navier-Stokes equations for a mesh point distribution sufficiently refined to represent turbulent motion are not feasible with current and projected computer systems (storage and processing time requirements); however, solutions of these equations with suitable subgrid scale turbulence modeling is currently feasible and should provide insight into the mechanics of turbulent flow. As indicated in figure 46, such

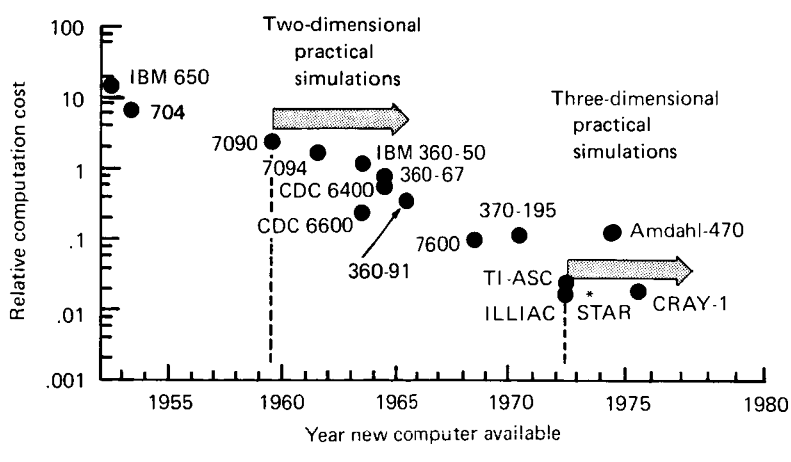


Figure 45.—Trend of computation cost for computer simulation of a given flow.

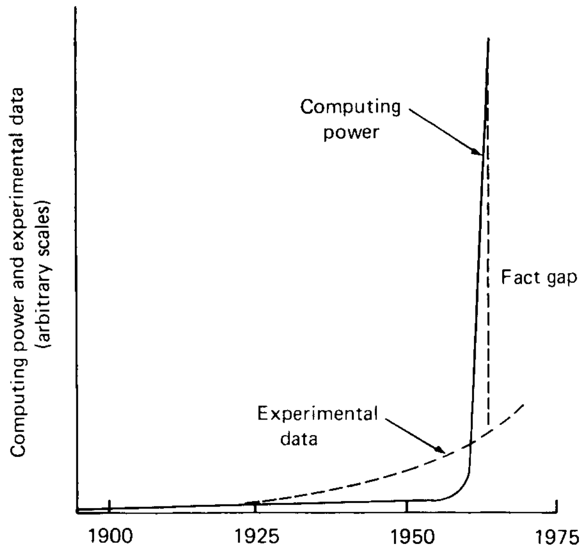


Figure 47.–Data and computers from Bradshaw (ref. 2).

solutions would require an increase in computational soeed of three orders of magnitude–an increase that appears to be feasible in view of current developments in computer technology and numerical procedures.

The increasing costs of developing and maintaining computer codes for numerical simulation currently tend to offset the decreasing cost of computer processing; consequently, each research objective must be carefully analyzed to assure optimum use of available resources, both manpower and computer systems, if accurate and efficient computer codes are to be developed that will have an impact on practical fluid mechanics problems. At the same time, we must insure that the right kind and amount of experimental data are obtained to verify and to provide necessary empirical inputs to the numerical simulation to minimize Professor Bradshaw's "fact gap" (fig. 47).

REFERENCES

1. von Neumann, John: On the Principles of Large Scale Computing Machines. Collected Works. Vol. V. Macmillan Co., 1963.
2. Bradshaw, Peter: The Understanding and Prediction of Turbulent Flow [Sixth Reynolds-Prandtl Lecture]. Aeronaut. J., vol. 76, July 1972, pp. 403-418.
3. AGARD: Computational Fluid Dynamics. Lecture Series 86, 1977.
4. Harris, Julius E.; and Morris, Dana J.: Solution of the Three-Dimensional Compressible, Laminar and Turbulent Boundary Layer Equations With Comparisons to Experimental Data. Proc. Int. Conf. Numer. Methods Fluid Dyn. 4th. Vol. 35 of Lecture Notes in Physics, Springer Verlag, 1975.
5. Kendall, Robert M.; Bonnett, William S.; Nardo, Charles T.; and Abbett, Michael J.: Three-Dimensional Compressible Boundary Layers of Reacting Gases Over Realistic Configurations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 77-99.
6. Cleary, Joseph W.: Effects of Angle of Attack and Bluntness on Laminar Heating-Rate Distributions of a 15° Cone at a Mach Number of 10.6. NASA TN D-5450, Oct. 1969.
7. Deiwert, G. S.: Computation of Separated Transonic Turbulent Flows. AIAA Paper 75-829, 1975.
8. Marvin, J. G.; Horstman, C. C.; Rubesin, M. W.; Coakley, T. J.; and Kussoy, M. I.: An Experimental and Numerical Investigation of Shock-Wave Induced Turbulent Boundary-Layer Separation at Hypersonic Speeds. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 377-400.
9. Mateer, George G.; Brosh, Aviel; and Viegas, John R.: An Experimental and Numerical Investigation of Normal Shock Wake Turbulent Boundary-Layer Interaction at $M_\infty = 1.5$. AIAA Paper 75-161, 1975.
10. Presley, Leroy L.: A Comparison of a Shock-Capturing Technique With Experimental Data for Three-Dimensional Internal Flows. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, pp. 623-642.
11. Erdos, John; Alzner, Edgar; Kalben, Paul; McNally, William; and Slutsky, Simon: Time Dependent Transonic Flow Solutions for Axial Turbomachinery. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 587-621.
12. Bilwakesh, K. R.; Koch, C. C.; and Prince, D. C.: Evaluation of Range and Distortion Tolerance for High Mach Number Transonic Fan Stages. Final Report on Task II: Performance of a 1500 Feet Per Second Tip Speed Transonic Fan Stage With Variable Geometry Inlet Guide Vanes and Stator. NASA CR-72880, 1972.
13. Salas, M. D.: The Anatomy of Floating Shock Fitting. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975, pp. 47-54.
14. Dash, Sanford M.; and Del Guidice, Paul D.: Numerical Methods for the Calculation of Three-Dimensional Nozzle Exhaust Flow Fields. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 659-701.
15. Hirsh, Richard S.: Calculation of Supersonic Three-Dimensional Free-Mixing Flows Using the Parabolic-Elliptic Navier-Stokes Equations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 543-565.
16. Barger, Raymond L.; and Brooks, Cuyler W., Jr.: A Streamline Curvature Method for Design of Supercritical and Subcritical Airfoils. NASA TN D-7770, 1975.
17. McGhee, Robert J.; and Beasley, William D.: Effects of Thickness on the Aerodynamic Characteristics of an Initial Low-Speed Family of Airfoils for General Aviation Applications. NASA TM X-72843, June 1976.
18. Olson, L. E.; and Dvorak, F. A.: Viscous/Potential Flow About Multi-Element Two-Dimensional and Infinite Span Swept Wings: Theory and Experiment. NASA TMX-62513, Dec. 1975.
19. Morgan, Harry L., Jr.: A Computer Program for the Analysis of Multielement Airfoils in Two-Dimensional Subsonic, Viscous Flow. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 713-748.
20. Goradia, S. H.; and Lilley, D. E.: Theoretical and Experimental Study of a New Method for Prediction of Profile Drag of Airfoil Sections. NASA CR-2539, 1975.
21. Garabedian, Paul R.: Computational Transonics. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1269-1280.
22. Jameson, A.: Transonic Potential Flow Calculations Using Conservative Form. Proc. AIAA Computational Fluid Dyn. 2d (Hartford, Conn.), June 19-20, 1975, pp. 148-161.
23. Murman, E.; and Cole, J.: Calculation of Plane Steady Transonic Flows. AIAA J., vol. 9, 1971, pp. 114-121.
24. Bavitz, R.: Analysis Method for Two-Dimensional Transonic Viscous Flow. NASA TN D-7718, 1974.
25. Bauer, F.; Garabedian, P.; and Korn, D.: A Theory of Supercritical Wing Sections, With Computer Programs and Examples. Vol. 66 of Lecture Notes in Econom-

ics and Mathematical Systems, M. Beckmann and H. P. Kunzi, eds., Springer Verlag, 1972.

26. Hicks, Raymond; Murman, Earll M.; and Vanderplaats, Garret N.: An Assessment of Airfoil Design by Numerical Optimization. NASA TM X-3092, 1974.
27. Vanderplaats, Garret N.; Hicks, Raymond; and Murman, Earll M.: Application of Numerical Optimization Techniques to Airfoil Design. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, pp. 749-768.
28. Jameson, A.: Iterative Solution of Transonic Flow Over Airfoils and Wings, Including Flows at Mach 1. Commun. Pure Appl. Math., vol. 27, 1974, pp. 283-309.
29. Mehta, U. B.; and Lavan, Z.: Starting Vortex, Separation Bubbles, and Stall–A Numerical Study of Laminar Unsteady Flow Around an Airfoil. J. Fluid Mech., vol. 67, no. 2, pp. 227-256.
30. Prandtl, L.: Essentials of Fluid Dynamics. Hafner Pub. Co., Inc., 1952, p. 200.
31. Leonard, A.: Simulation of Unsteady Three-Dimensional Separated Flows With Interacting Vortex Filaments. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 925-937.
32. Achenbach, E.: Vortex Shedding From Spheres. J. Fluid Mech., vol. 62, 1974, p. 209.
33. Ehlers, F. Edward; Johnson, Forrester T.; and Rubbert, Paul E.: Advanced Panel-Type Influence Coefficient Methods Applied to Subsonic and Supersonic Flows. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 939-984.
34. Polhamus, Edward C.: A Concept of the Vortex Lift of Sharp-Edge Delta Wings Based on a Leading-Edge-Suction Analogy. NASA TN D-3767, 1966.
35. Weber, James A.; Brune, Guenter W.; Johnson, Forrester T.; Lu, Paul; and Rubbert, Paul E.: A Three-Dimensional Solution of Flows Over Wings With Leading Edge Vortex Separation. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1013-1032.
36. Lan, E.: Analytical Investigation of Wing Jet Interaction. NASA CR-138140, Apr. 1974.
37. Phelps, Arthur E.; Letko, William; and Henderson, Robert L.: Low-Speed Wind-Tunnel Investigation of a Semispan STOL Jet Transport Wing-Body With an Upper Surface Blown Jet Flap. NASA TN D-7183, May 1973.
38. Lopez, M. L.; and Shen, C. C.: Recent Developments in Jet Flap Theory and Its Application to STOL Aerodynamic Analysis. AIAA Paper No. 71-578, June 1971.
39. Keller, James D.; and South, Jerry C.: RAXBOD: A Fortran Program for Inviscid Transonic Flow Over Axisymmetric Bodies. NASA TM X-72831, Feb. 1976.
40. Fox, Charles H., Jr.: Experimental Surface Pressure Distributions for a Family of Axisymmetric Bodies at Subsonic Speeds. NASA TM X-2439, Dec. 1971.
41. Lomax, H.; Bailey, F. R.; and Ballhaus, W. F.: On the Numerical Simulation of Three-Dimensional Transonic Flow With Application to the C-141 Wing. NASA TN D-6933, 1973.
42. Bailey, F. R.; and Ballhaus, W. F.: Comparisons of Computed and Experimental Pressures for Transonic Flows About Isolated Wings and Wing-Fuselage Configurations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1213-1231.
43. Newman, Perry A.; and Klunker, E. B.: Numerical Modeling of Tunnel-Wall and Body-Shape Effects on Transonic Flows Over Finite Lifting Wings. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1189-1212.
44. Barnwell, Richard W.: Approximate Method for Calculating Transonic Flow About Lifting Wing-Body Combinations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1281-1303.
45. Rakich, John V.; and Kutler, Paul: Comparison of Characteristics and Shock Capturing Methods With Application to the Space Shuttle Vehicle. AIAA Paper No. 72-191, 1972.
46. Marconi, Frank; Yaeger, Larry; and Hamilton, H. Harris: Computation of High-Speed Inviscid Flows About Real Configurations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1411-1455.
47. Goodrich, W. D.; Li, C. P.; Houston, C. K.; Meyers, R. M.; and Olmedo, L.: Scaling of Orbiter Aerothermodynamic Data Through Numerical Flow Field Simulations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1395-1410.

BIBLIOGRAPHY

The following list of publications is a fairly comprehensive, although not all-inclusive, current state-of-the-art representation of computational aerodynamics in the United States. The list covers the most recent publications; references to earlier work in a given area will be found in the references that normally appear in the individual documents listed in this bibliography.

The list is not categorized by application or computational method because many papers are applicable to more than one category. Instead, the list is alphabetized by author as an aid in relating the names given in the figures of this paper to other publications.

Bailey, F. R.; and Ballhaus, W. F.: Comparisons of Computed and Experimental Pressures for Transonic Flows About Isolated Wings and Wing-Fuselage Configurations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1213-1231.

Baker, A. J.; Rogers, R. Clayton; and Zelazny, S. W.: Analytic Study of Mixing and Reacting Three-Dimensional Supersonic Combustor Flow Fields. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 251-315.

Baldwin, B. S.; and MacCormack, R. W.: Interaction of Strong Shock Wave With Turbulent Boundary Layer. Proc. Int. Conf. Numer. Methods Fluid Dyn. 4th. Vol. 35 of Lecture Notes in Physics, Springer Verlag, 1975.

Baldwin, B. S.; and MacCormack, R. W.: Numerical Solution of the Interaction of a Strong Shock Wave With a Hypersonic Turbulent Boundary Layer. AIAA Paper 74-558, AIAA 7th Fluid and Plasma Dyn. Conf. (Palo Alto), June 17-19, 1974.

Baldwin, B. S.; MacCormack, R. W.; and Deiwert, G. S.: Numerical Techniques for the Solution of the Compressible Navier Stokes Equations and Implementation of Turbulence Models. Computational Methods for Inviscid and Viscous Two- and Three-Dimensional Flow Fields. Lecture Series 73, AGARD-LSP-73, 1975.

Baldwin, B. S.; and Rose, W. C.: Calculation of Shock-Separated Turbulent Boundary Layers. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 401-417.

Ballhaus, W. F.; and Bailey, F. R.: Numerical Calculation of Transonic Flow About Swept Wings. AIAA Paper 72-677, June 1972.

Barger, Raymond L.: Procedures for the Design of Low Pitching Moment Airfoils. NASA TN D-7982.

Barger, Raymond L.; and Brooks, Cuyler W., Jr.: A Streamline Curvature Method for Design of Supercritical and Subcritical Airfoils. NASA TN D-7770.

Barnwell, Richard W.: Approximate Method for Calculating Transonic Flow About Lifting Wing-Body Combinations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1281-1303.

Bauer, F.; Garabedian, P.; and Korn, D.: A Theory of Supercritical Wing Sections, With Computer Programs and Samples. Vol. 66 of Lecture Notes in Economics and Mathematical Systems, M. Backmann and H. P. Kunzi, eds., Springer Verlag, 1972.

Bauer, F.; Garabedian, P.; Korn, D.; and Jameson, A.: Supercritical Wing Sections II. Vol. 108 of Lecture Notes in Economics and Mathematical Systems, Springer Verlag, 1975.

Bauer, F.; and Korn, D.: Computer Simulation of Transonic Flow Past Airfoils With Boundary Layer Correction. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Bavitz, R.: Analysis Method for Two-Dimensional Transonic Viscous Flow. NASA TN D-7718, 1974.

Beam, Richard M.; and Ballhaus, William F.: Numerical Integration of the Small-Disturbance Potential and Euler Equations for Unsteady Transonic Flow. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 789-809.

Bland, Samuel R.: Recent Advances and Concepts in Unsteady Aerodynamics Theory. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1305-1326.

Blottner, F. G.: Computational Techniques for Boundary Layers. Computational Methods for Inviscid and Viscous Two- and Three-Dimensional Flow Fields. Lecture Series 73, AGARD-LSP-73, 1975.

Buckingham, A. C.; and Birnbaum, N. K.: Three Dimensional Compressible Flow Over a Rigid Structure: Explicit Finite Difference and Integral Method Coupled at Slip Walls. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Bushnell, D.; and Beckwith, I.: AIAA Paper No. 69-684, 1969.

Carlson, L. A.: Transonic Airfoil Analysis and Design Using Cartesian Coordinates. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Carter, J. E.: Numerical Solutions of the Supersonic Laminar Flow Over a Two-Dimensional Compression Corner. Vol. 19 of Lecture Notes in Physics, Springer Verlag, 1973, p. 69.

Carter, James E.; and Wornom, Stephen F.: Solutions for Incompressible Separated Boundary Layers Including Viscous-Inviscid Interaction. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 125-150.

Cebeci, T.; et al.: Some Problems of the Calculation of Three Dimensional Boundary Layer Flows on General Considerations. NASA CR-2285, 1973.

Cleary, Joseph W.: Effects of Angle of Attack and Bluntness on Laminar Heating-Rate Distributions of a 15° Cone at a Mach Number of 10.6. NASA TN D-5450, Oct. 1969.

Dash, Sanford M. and Del Guidice, Paul D.: Numerical Methods for the Calculation of Three-Dimensional Nozzle Exhaust Flow Fields. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 659-701.

Davy, W. C.; and Reinhardt, W. A.: Computation of Shuttle Nonequilibrium Flow Fields on a Parallel Processor. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1351-1376.

Deardorff, J. W.: A Numerical Study of Three-Dimensional Turbulent Channel Flow at Large Reynolds Numbers. J. Fluid Mech., vol. 41, 1970, p. 453.

Deiwert, G. S.: Computation of Separated Transonic Turbulent Flows. AIAA Paper 75-829, 1975.

Deiwert, G. S.: High Reynolds Number Transonic Flow Simulation. Proc. Int. Conf. Numer. Methods Fluid Dyn. 4th. Vol. 35 of Lecture Notes in Physics, Springer Verlag, 1975.

Deiwert, G. S.: Numerical Simulation of High Reynolds Number Transonic Flow. AIAA Paper 74-603, AIAA 7th Fluid and Plasma Dyn. Conf. (Palo Alto), June 17-19, 1974.

Deiwert, George S.; McDevitt, John B.; and Levy, Lionel L., Jr.: Simulation of Turbulent Transonic Separated Flow Over an Airfoil. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 419-436.

Der, J., Jr.; and Raetz, G. S.: Solution of General Three-Dimensional Laminar Boundary Layer Problems by an Exact Numerical Method. IAS Paper 62-70, Jan. 1962.

Donaldson, C. du P.: Calculation of Turbulent Shear Flows for Atmospheric and Vortex Motions. IAAA Paper 71-217, AIAA 9th Aerospace Sciences Meeting (New York), Jan. 25-27, 1971.

Dwyer, H. A.: Hypersonic Boundary Layer Studies on a Spinning Cone at Angle of Attack. AIAA Paper 71-57, Jan. 1971.

Dwyer, H. A.; and Sanders, B. R.: A Physically Optimum Difference Scheme for Three-Dimensional Boundary Layers. Proc. Int. Conf. Numer. Methods Fluid Dyn. 4th. Vol 35 of Lecture Notes in Physics, Springer Verlag, 1975, pp. 144-150.

Ehlers, F. Edward; Johnson, Forrester T.; and Rubbert, Paul E.: Advanced Panel-Type Influence Coefficient Methods Applied to Subsonic and Supersonic Flows. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 939-984.

Erdos, John; Alzner, Edgar; Kalben, Paul; McNally, William; and Slutsky, Simon: Time-Dependent Transonic Flow Solutions for Axial Turbomachinery. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 587-621.

Fannelop, T. K.: A Method of Solving the Three-Dimensional Laminar Boundary-Layer Equations With Application to a Lifting Re-Entry Body. AIAA J., vol. 6, no. 6, June 1969, pp. 1075-1084.

Frieders, Michael C.; and Lewis, Clark H.: Effects of Mass Transfer Into Laminar and Turbulent Boundary Layers Over Cones at Angle of Attack. Arnold Engineering Development Center TR 75-55, 1975.

Garabedian, Paul R.: Computational Transonics. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1269-1280.

Garabedian, Paul R.: Partial Differential Equations. John Wiley & Sons, Inc., 1964.

Goodrich, W. D.; Li, C. P.; Houston, C. K.; Meyers, R. M.; and Olmedo, L.: Scaling of Orbiter Aerothermodynamic Data Through Numerical Flow Field Simulations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1395-1410.

Goradia, S. H.; and Lilley, D. E.: Theoretical and Experimental Study of a New Method for Prediction of Profile Drag of Airfoil Sections. NASA CR-2539, 1975.

Hall, M. G.: A Numerical Method for Calculating Steady Three-Dimensional Laminar Boundary Layers. TR 67145, Royal Aircraft Establishment (Farnborough, United Kingdom), June 1967.

Harris, Julius E.; et al.: Calculation of Three-Dimensional Compressible Laminar and Turbulent Boundary Layers. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 19-99.

Harris, W. L.; and Kumar, S.: Computational Aspects of Non-Similar Solutions of the Rarefied Leading Edge on a Cone. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Harris, Julius E.; and Morris, Dana J.: Solution of the Three-Dimensional Compressible, Laminar and Turbulent Boundary Layer Equations With Comparisons to Experimental Data. Proc. Int. Conf. Numer. Methods Fluid Dyn. 4th. Vol. 35 of Lecture Notes in Physics, Springer Verlag, 1975.

Hirsh, Richard S.: Calculation of Supersonic Three-Dimensional Free-Mixing Flows Using the Parabolic-Elliptic Navier Stokes Equations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 543-565.

Holden, M. S.: Shock Wave-Turbulent Boundary Layer Interaction in Hypersonic Flow. AIAA Paper 72-74, AIAA 10th Aerospace Sciences Meeting (San Diego), Jan. 1972.

Holst, Terry L.; Tannehill, John C.; and Rakich, John V.: Numerical Computation of Viscous Blunt Body Flows With a Planar Impinging Shock. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1457-1471.

Hung, C. M.; and MacCormack, R. W.: Numerical Solutions of Supersonic and Hypersonic Laminar Flows Over a Two Dimensional Compression Corner. AIAA Paper 75-2 (Pasadena, Calif.), Jan. 1975.

Inouye, M.; Marvin, J. G.; and Sheaffer, Y. S.: Turbulent-Wake Calculations With an Eddy-Viscosity Model. AIAA J., vol. 10, no. 2, Feb. 1972, pp. 216-217.

Isaacson, E.; and Keller, H. B.: Analysis of Numerical Methods. John Wiley & Sons, Inc., 1966.

Jameson, A.: Iterative Solution of Transonic Flow Over Airfoils and Wings, Including Flows at Mach 1. Commun. Pure Appl. Math., vol. 27, 1974, pp. 283-309.

Jameson, A.: Transonic Potential Flow Calculations Using Conservative Form. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975, pp. 148-161.

Jones, D.: AGARDograph 137, 1971.

Kendall, R. M.: An Analysis of the Coupled Chemically Reacting Boundary Layer and Charring Ablator. Pt. V of A General Approach to the Thermodynamic Solution of Mixed Equilibrium-Non Equilibrium, Homogeneous or Heterogeneous Systems. NASA CR-1064, 1968.

Kendall, R. M.; Barnett, W. S.; Nardo, C. T.; and Abbett, M. J.: Computational Procedure for Three-Dimensional Boundary Layers on Aircraft and Aerospace Vehicles. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Kitchens, C. W., Jr.; Gerber, N.; and Sedney, R.: "Computational Implications of the Zone of Dependence Concept for Three-Dimensional Boundary Layers on a Spinning Body. Ballistic Res. Lab. Rept. No. 1774, Department of the Army, Apr. 1975.

Kitchens, C. W., Jr.; Sedney, R.; and Gerber, N.: The Role of the Zone of Dependence Concept in Three Dimensional Boundary Layer Calculations. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Krause, E.: Comment on Solution of a Three-Dimensional Boundary Layer Flow With Separation. AIAA J., vol. 7, no. 3, Mar. 1969, pp. 575-576.

Kutler, Paul: Computation of Three-Dimensional Inviscid Supersonic Flows. Lecture Notes In Physics, Springer Verlag, 1974.

Kutler, P.; Reinhardt, W. A.; and Warming, R. F.: Numerical Computation of Multishocked Three-Dimensional Supersonic Flow Fields With Real Gas Effects. AIAA Paper 72-702, 1972.

Kutler, Paul; and Sakell, Leonidas: Three-Dimensional, Shock-on-Shock Interaction Problem. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1111-1140.

Kutler, Paul; Shankar, Vijaya; Anderson, Dale A.; and Sorenson, Rease L.: Internal and External Axial Corner Flow. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 643-658.

Lamar, John E.: Some Recent Applications of the Suction Analogy to Vortex-Lift Estimates. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 985-1011.

Lan, E.: Analytical Investigation of Wing Jet Interaction. NASA CR-138140, Apr. 1974.

Leonard, A.: Simulation of Unsteady Three-Dimensional Separated Flows With Interacting Vortex Filaments. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 925-937.

Lin, C. L.; Pepper, D. W.; and Lee, S. C.: Numerical Methods for Separated Flow Solutions Around a Circular Cylinder. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Lomax, H.: Recent Progress in Numerical Techniques for Flow Simulation. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Loos, H. G.: A Simple Laminar Boundary Layer With Secondary Flow. J. Aerosp. Sci., vol. 22, no. 1, 1955, pp. 35-40.

Lung, J. L.; Tiegerman, B.; Yu, N. J.; and Seebass, A. R.: Advances in Sonic Boom Theory. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1033-1047.

MacCormack, R. W.: Numerical Solution of the Interaction of a Shock Wave With a Laminar Boundary Layer. Vol. 8 of Lecture Notes in Physics, Springer Verlag, 1971, p. 151.

MacCormack, R. W.: The Effect of Viscosity in Hypervelocity Impact Cratering. AIAA Paper 69-354, 1969.

MacCormack, R. W.; and Paullay, A. J.: Computational Efficiency Achieved by Time Splitting of Finite Difference Operators. AIAA Paper 72-154, 1972.

McGhee, Robert J.; Beasley, William D.; and Somers, Dan M.: Low Speed Aerodynamic Characteristics of a 13-Percent-Thick Airfoil Section Designed for General Aviation Applications. NASA TM X-72697, 1975.

McGowan, J. J., II; and Davis, R. T.: Development of a Numerical Method to Solve the Three-Dimensional Compressible Laminar Boundary-Layer Equations With Application to Elliptical Cones at Angle of Attack. ARL 70-0341, Aerospace Research Laboratory, Dec. 1970.

Mack, Leslie M.: A Numerical Method for the Prediction of High-Speed Boundary-Layer Transition Using Linear Theory. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 101-123.

Marconi, Frank; Yaeger, Larry; and Hamilton, H. Harris: Computation of High-Speed Inviscid Flows About Real Configurations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1411-1455.

Margason, R. J.; Yip, Long P.; and Gainer, Thomas G.: Recent Developments in Propulsive-Lift Aerodynamic Theory. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 871-895.

Martin, E. D.: A Fast Semi-Direct Method for Computing Transonic Aerodynamic Flows, Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Martin, E. Dale: Progress in Application of Direct Elliptic Solvers to Transonic Flow Computations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 839-870.

Marvin, J. G.; Horstman, C. C.; Rubesin, M. W.; Coakley, T. J.; and Kussoy, M. I.: An Experimental and Numerical Investigation of Shock-Wave Induced Turbulent Boundary-Layer Separation at Hypersonic Speeds. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 377-399.

Marvin, J. G.; and Sheaffer, Y. S.: A Method for Solving the Nonsimilar Laminar Boundary-Layer Equations Including Foreign Gas Injection. NASA TN D-5516, Nov. 1969.

Mateer, George G.; Brosh, Aviel; and Viegas, John R.: An Experimental and Numerical Investigation of Normal Shock Wave Turbulent Boundary-Layer Interaction at $M_\infty = 1.5$. AIAA Paper 75-161, 1975.

Mehta, U. B.; and Lavan, Z.: Starting Vortex, Separation Bubbles, and Stall–A Numerical Study of Laminar Unsteady Flow Around an Airfoil. J. Fluid Mech., vol. 67, no. 2, 1975, pp. 227-256.

Melnick, R. E.; and Chow, R.: Asymptotic Theory of Two-Dimensional Trailing-Edge Flows. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 177-249.

Miller, David S.; and Middleton, Wilbur D.: An Integrated System for the Aerodynamic Design and Analysis of Supersonic Aircraft. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1049-1065.

Moretti, G.: A Circumspect Exploration of a Difficult Feature of Multidimensional Imbedded Shocks. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Morino, Luigi; and Chen, Lee-Tzong: Indicial Compressible Potential Aerodynamics Around Complex Aircraft Configurations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1067-1110.

Murman, E.: Analysis of Embedded Shock Waves Calculated by Relaxation Methods. Paper presented at AIAA Computational Fluid Dyn. Conf. (Palm Springs, Calif.), 1973.

Murman, E.; and Cole, J.: Calculation of Plane Steady Transonic Flows. AIAA J., vol. 9, 1971, pp. 114-121.

Murphy, John D.; Presley, Leroy L.; and Rose, W. C.: On the Calculation of Supersonic Separating and Reattaching Flows. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 151-175.

Nachtsheim, P. R.; and Swigert, P.: Satisfaction of Asymptotic Boundary Conditions in Numerical Solutions of Systems of Nonlinear Equations of the Boundary-Layer Type. NASA TN D-3004, 1965.

Nash, J.; and Macdonald, A.: The Calculation of Momentum Thickness in a Turbulent Boundary Layer at Mach Numbers Up to Unity. C.P. No. 963, Aerosp. Research Council, London, 1967.

Newman, Perry A.; and Klunker, E. B.: Numerical Modeling of Tunnel-Wall and Body-Shape Effects on Transonic Flows Over Finite Lifting Wings. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1189-1212.

O'Brien, G. G.; Hyman, M. A.; and Kaplan, S.: A Study of the Numerical Solution of Partial Differential Equations. J. Math. Phys., vol. 29, 1950, pp. 223-251.

Oh, Youn H.; and Bushnell, Dennis M.: Influence of External Disturbances and Compressibility on Free Turbulent Mixing. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 341-376.

Oliver, David A.; and Sparis, Panagiotis: Computational Aspects of the Prediction of Multidimensional Transonic Flows in Turbomachinery. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 567-585.

Orszag, S. A.: Numerical Simulation of Boundary Layer Transition on a Flat Plate. Flow Research Rept. No. 33, Flow Research, Inc., May 1974.

Presley, Leroy L.: A Comparison of a Shock-Capturing Technique With Experimental Data for Three-Dimensional Internal Flows. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 623-642.

Rakich, John V.; and Lubard, Stephen C.: Numerical Computation of Viscous Flows on the Lee Side of Blunt Shapes Flying at Supersonic Speeds. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 531-542.

Rakich, John V.; and Pegot, Eva B.: Flow Field and Heating on the Windward Side of the Space Shuttle Orbiter. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1377-1394.

Rizzi, Arthur W.; and Bailey, Harry E.: Reacting Nonequilibrium Flow Around the Space Shuttle Using a Time-Split Method. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1327-1349.

Rizzi, A. W.; and Bailey, H. E.: A Generalized Hyperbolic Marching Method for Chemically Reacting 3-D Supersonic Flow Using a Splitting Technique. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Rogallo, R. S.: Discrete Vortex Simulation of Two-Dimensional Incompressible Separated Flows. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Rosenhead, Laminar Boundary Layers. Oxford Univ. Press, Inc., 1963, p. 464.

Rossow, Vernon J.: Survey of Computational Methods for Lift-Generated Wakes. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 897-923.

Rossow, V. J.: Theoretical Study of Lift-Generated Vortex Wakes Designed to Avoid Rollup. AIAA J., vol. 13, no. 4, Apr. 1975, pp. 476-484.

Rubesin, Morris W.: Subgrid- or Reynolds Stress-Modeling for Three-Dimensional Turbulence Computations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 317-339.

Rubesin, M. W.; and Rose, W. C.: The Turbulent Mean-Flow, Reynolds-Stress, and Heat-Flux Equations in Mass-Averaged Dependent Variables. NASA TM X-62248, Mar. 1973.

Rubin, Stanley G.; and Graves, Randolph A., Jr.: Viscous Flow Solutions With a Cubic Spline Approximation. Vol. 3 of Computers and Fluids, Pergamon Press, 1975, pp. 1-36.

Rubin, S. G.; and Khosla, P. K.: Higher Order Numerical Solutions Using Cubic Splines. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Rudy, David H.; Morris, Dana J.; Blanchard, Doris K.; Cooke, Charlie H.; and Rubin, Stanley G.: An Investigation of Several Numerical Procedures for Time-Asymptotic Compressible Navier-Stokes Solutions. Aerodynamic Analyses

Requiring Advanced Computers. NASA SP-347, 1975, pp. 437-468.

Ruo, S. Y.; and Theisen, J. G.: Calculation of Unsteady Transonic Aerodynamics for Oscillating Wings With Thickness. NASA CR-2259, 1975.

Saffman, P. G.; and Wilcox, D. C.: Turbulence Model Predictions for Turbulent Boundary Layers. AIAA J., vol. 12, no. 4, 1974.

Salas, M. D.: The Anatomy of Floating Shock Fitting. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975, pp. 47-54.

South, Jerry C., Jr.; and Keller, James D.: Axisymmetric Transonic Flow Including Wind-Tunnel Wall Effects. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1233-1267.

Sowerby, L., Secondary Flow in a Boundary Layer, Aerospace Research Council Rept. 16832, London, 1954.

Steger, Joseph L.; and Lomax, Harvard: Calculation of Inviscid Shear Flow Using a Relaxation Method for the Euler Equations. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 811-838.

Thames, Frank C.; Thompson, Joe F.; and Mastin, C. Wayne: Numerical Solution of the Navier-Stokes Equations for Arbitrary Two-Dimensional Airfoils. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 469-530.

Thomas, P. D.; and Wilson, K. H.: Efficient Computation of "Stiff" Chemically Reacting Flow in Turbulent Free Jets. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Thompson, J. F.; Thames, F. C.; Mastin, C. W.; and Shanks, S. P.: Use of Numerically Generated Body-Fitted Coordinate Systems for Solution of the Navier Stokes Equations. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Townsend, A. A.: Equilibrium Layers and Wall Turbulence. J. Fluid Mech., vol. 11, 1961, pp. 97-120.

Townsend, A. A.: The Structure of Turbulent Shear Flow. Cambridge Univ. Press, 1956.

Tracy, R. R.: Hypersonic Flow Over a Yawed Circular Cone. GAL CIT Memo. No. 69, Calif. Inst. Technology, Aug. 1963.

Vanderplaats, Garret N.; Hicks, Raymond; and Murman, Earll M.: Application of Numerical Optimization Techniques to Airfoil Design. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 749-768.

Wang, K. C.: Three-Dimensional Laminar Boundary Layer Over a Body of Revolution at Incidence. Part VI: General Methods and Research of the Case at High Incidence. MML TR 73-02c, Martin Marietta Lab., May 1973.

Wang, K. C.: An Effective Approximation for Computing the Three-Dimensional Laminar Boundary Layer Flows. AIAA J., vol. 9, no. 8, Aug. 1971, pp. 1649-1651.

Warming, R. F.; and Beam, R. M.: Upwind Second Order Difference Schemes and Applications in Unsteady Aerodynamic Flows. Proc. AIAA Computational Fluid Dyn. Conf. 2d (Hartford, Conn.), June 19-20, 1975.

Warming, R. F.; and Hyett, B. J.: The Modified Equation Approach to the Stability and Accuracy Analysis of Finite-Difference Methods. J. Comp. Physics., vol. 14, no. 2, Feb. 1974.

Weber, James A.; Brune, Guenter W.; Johnson, Forrester, T.; Lu, Paul; and Rubbert, Paul E.: A Three-Dimensional Solution of Flows Over Wings With Leading Edge Vortex Separation. Aerodynamic Analyses Requiring Advanced Computers. NASA SP-347, 1975, pp. 1013-1047.

Whitcomb, R.: Review of NASA Supercritical Airfoil. ICAS Paper No. 74-10, Congr. Int. Council Aeronaut. Sci. 9th (Haifa, Israel), Aug. 25-30, 1974.

Wilcox, D. C.: Calculation of Turbulent Boundary Layer Shock Wave Interaction. AIAA J., vol. 11, no. 4, 1973, pp. 1592-1594.

Wilcox, D. C.; and Alber, I. E.: A Turbulence Model for High Speeds. Proceedings of the 1972 Heat Transfer and Fluid Mechanics Institute, Stanford Univ. Press, June 1972.

Wu, J. C.: A Flowfield Segmentation Method for the Numerical Solution of Viscous Flow Problems. Proc. Int. Conf. Numer. Methods Fluid Dyn. 4th. Vol. 35 of Lecture Notes in Physics, Springer Verlag, 1975.

Yohner, P. L.; and Hansen, A. G.: Some Numerical Solutions of Similarity Equations for Three-Dimensional Laminar Incompressible Boundary Layer Flows. NACA TN 4370, 1958.

NASA-Langley, 1977